

InfineonSpice

2026.2

About this document

This guide describes how to use InfineonSpice, version 2026.2 for developing circuits, defining simulations, and viewing the results.

Intended audience

This document is meant to support any users of InfineonSpice.

Table of contents

	About this document	1
	Table of contents	1
1	Installing InfineonSpice	4
1.1	General Information	4
1.1.1	System Requirements	4
1.1.2	Download the Software	4
1.2	Installation on Windows	5
1.2.1	Quick Installation	5
1.2.2	Custom Installation	9
1.2.3	Uninstall	19
2	Getting started	20
2.1	Introduction	21
2.2	Creating a project	21
2.3	Creating a schematic	22
2.4	Setting up a simulation	29
2.5	Running a simulation	32
3	Using InfineonSpice	34
3.1	Work environment	34
3.1.1	Values and expressions	35
3.1.2	Keyboard shortcuts	36
3.1.3	View Options	39
3.1.4	Tips & Tricks	41
3.2	Project structure	42
3.3	Setting up a project	43
3.3.1	Creating a new project	43
3.3.2	Setting up new projects	44
3.3.3	Managing project contents	45
3.3.4	Managing Infineon and third-party components	48

Table of contents

3.3.4.1	Working with the Download Manager	49
3.4	Working with schematics	51
3.4.1	Creating a new schematic	52
3.5	Working with symbols	54
3.5.1	Creating a new symbol	55
3.6	Creating a local component	58
3.6.1	Working with libraries	61
3.7	Setting up a simulation	62
3.7.1	Defining simulation profiles	63
3.7.2	Setting up a transient analysis	65
3.7.3	Setting up an operating point analysis	66
3.7.4	Setting up a DC analysis	66
3.7.5	Setting up an AC analysis	68
3.8	Running a simulation	69
3.8.1	Output files of a simulation	70
3.8.2	Running InfineonSpice from the command line	71
3.9	Evaluating simulations	72
3.9.1	Working with plots	73
3.9.2	Working with traces	74
3.9.3	Working with axes	76
3.9.4	Working with annotations	78
3.9.5	Evaluating AC analyses	78
3.10	Data sharing	80
3.10.1	Custom containers	80
3.10.2	Exporting a project	81
3.10.3	Importing a project	82
4	InfineonSpice reference	83
4.1	Naming directories and files	83
4.2	Menus	83
4.2.1	File menu	83
4.2.1.1	Import Project dialog	85
4.2.1.2	Print Preview window	86
4.2.2	Edit menu	88
4.2.2.1	Schematic Page Setup dialog	92
4.2.2.2	Properties dialog	94
4.2.2.3	Attributes dialog	96
4.2.2.4	Substitution Field dialog	97
4.2.2.5	Text Attributes dialog	98
4.2.2.6	Net color dialog	100
4.2.2.7	Net Colors dialog	101
4.2.3	View menu	101
4.2.4	Place menu (schematics tab)	102

Table of contents

4.2.4.1	Place Text dialog	104
4.2.5	Place menu (symbol tab)	106
4.2.6	Options menu	107
4.2.6.1	Keyboard Shortcuts dialog	108
4.2.6.2	Display Options dialog	110
4.2.6.3	Select Font dialog	114
4.2.7	Window menu	115
4.2.8	Help menu	116
4.3	Windows and views	117
4.3.1	Work area	118
4.3.2	Workspace view	118
4.3.2.1	Project menu	120
4.3.2.2	Schematic menu	122
4.3.2.3	Library menu	122
4.3.2.4	Symbol menu	123
4.3.3	Model Libraries view	123
4.3.4	Simulation Environment view	126
4.3.5	Properties	129
4.3.6	Output	129
4.4	Waveform viewer	129
4.4.1	File	130
4.4.2	View	132
4.4.3	Plot	133
4.4.4	Trace	137
4.4.5	Tools	138
4.4.6	Help	140
5	Glossary	141
6	Support and feedback	142
	Disclaimer	143

1 Installing InfiniteonSpice

This chapter gives an overview of the installation of InfiniteonSpice. It contains information about system requirements and where to download the software as well as instructions for a quick or custom installation.

1.1 General Information

This chapter gives an overview of the system requirements as well as information where to download the latest version of the software.

1.1.1 System Requirements

The following system requirements have to be fulfilled in order to use the InfiniteonSpice Software.

- Windows 10
- Minimum of 4 GB RAM
- Approximately 815MB of disk space
- Network connection for installation, licensing and content access

InfiniteonSpice does need a license to run - a default license is shipped with the installation. You need to be connected to the internet during the installation process and the first time you use the software in order to validate your license. After that it will be validated periodically.

Note: *The software is not supported on 32-bit operating systems.*

1.1.2 Download the Software

The latest Release of the InfiniteonSpice Software can be downloaded here: [Released Versions](#)

For the installation

1. Open the page in a browser,
2. Choose the latest version, download the installer and
3. Refer to chapter [Installation on Windows](#) for a detailed installation description

1 Installing InfiniteSpice

1.2 Installation on Windows

InfiniteSpice comes with a ready-to-use executable installer, which will guide you through the installation process. Once the installation is finished you will find InfiniteSpice in the Start Menu.

1.2.1 Quick Installation

This chapter explains the default installation of InfiniteSpice on your computer.

Run the InfiniteSpice_2026.2_Windows_x64.exe installer and follow the prompts.

First of all you can select the install mode. Choose if you want to install the software only for you (recommended) or for all users.

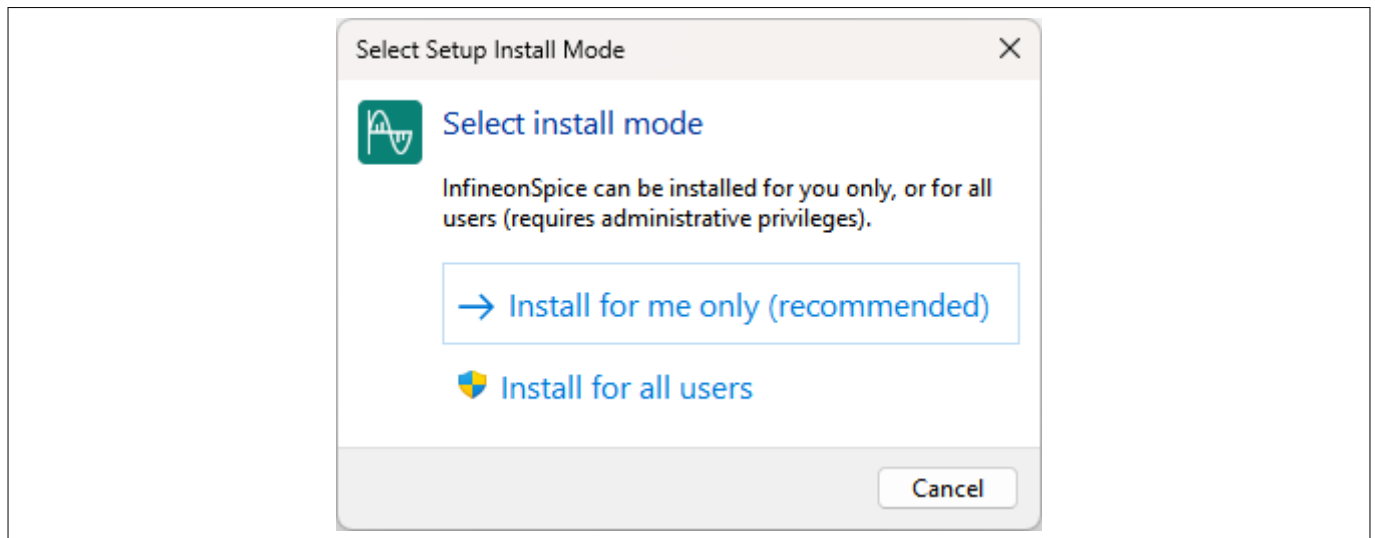


Figure 1 **Select install mode**

Read the license agreement document presented on the next page of the installation wizard ([Figure 2](#)) and select **I accept the agreement**. The installation cannot proceed without this. Click **Next** to proceed. Or click **Cancel** if you wish to close the installation wizard.

1 Installing InfineonSpice

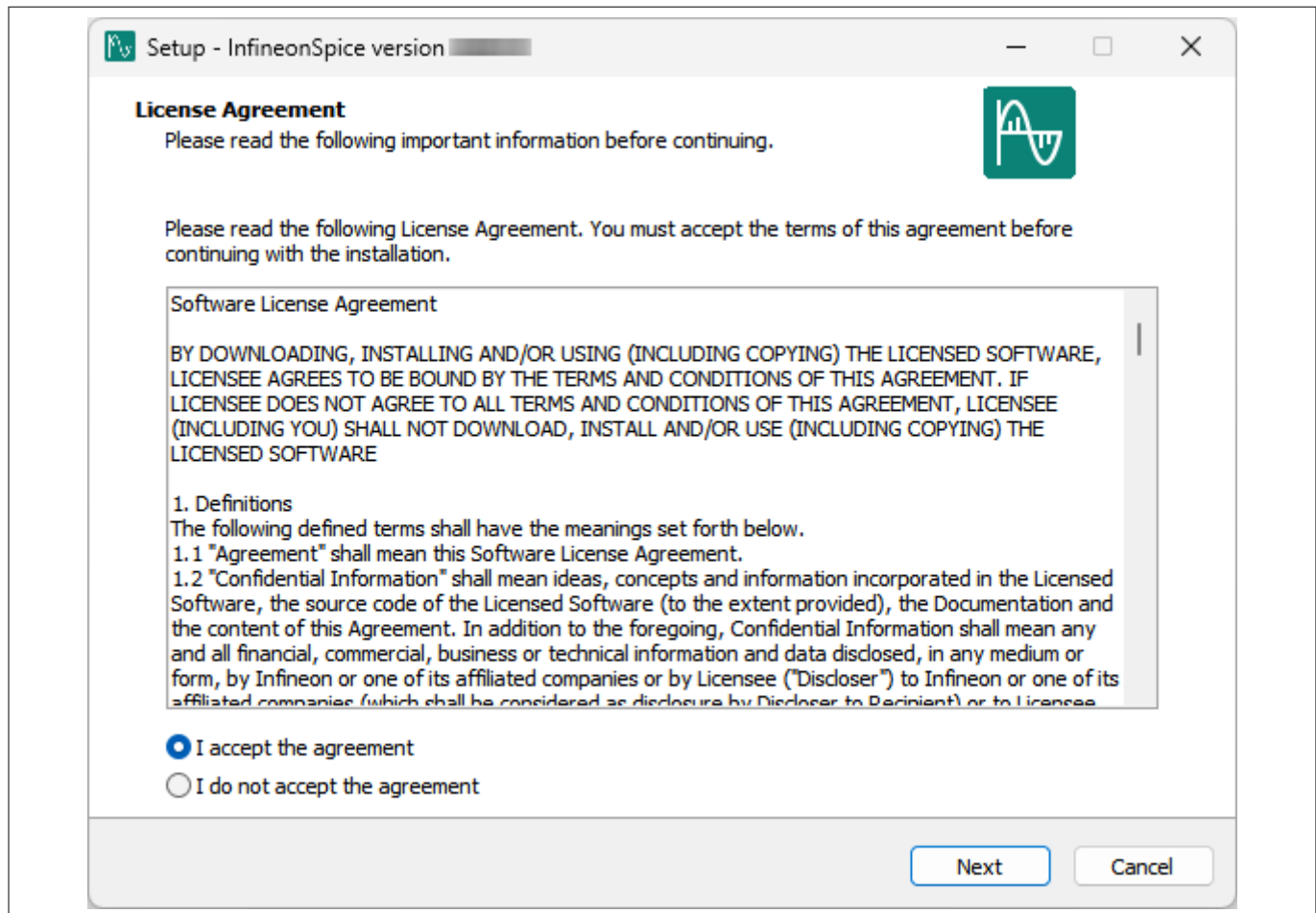


Figure 2 **License Agreement**

On the next page please choose **Quick Installation** and proceed with **Install**.

1 Installing InfiniteonSpice

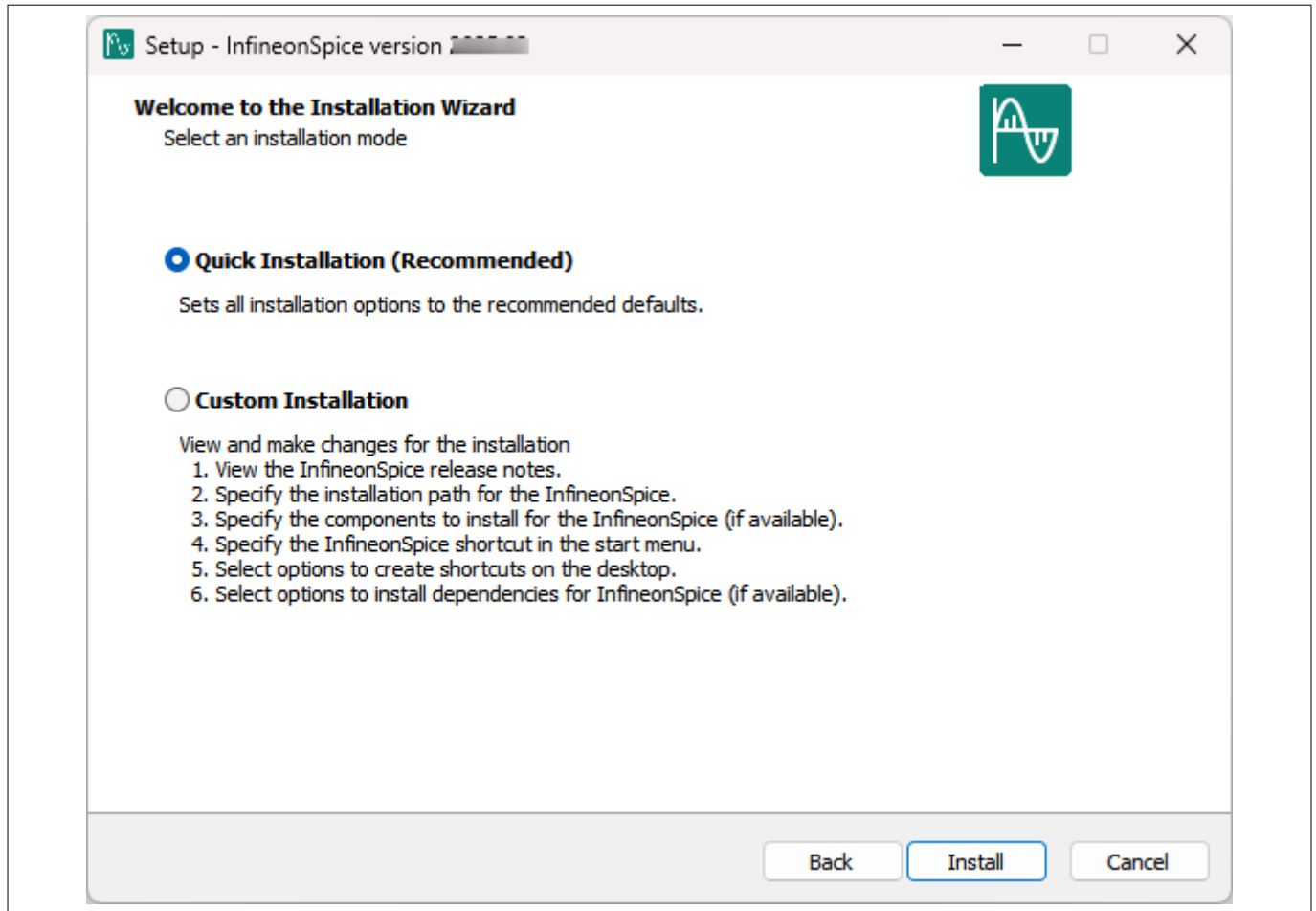


Figure 3 **Quick Installation**

The following wizard will display the progress of the installation ([Figure 4](#)).

1 Installing InfiniteSpice

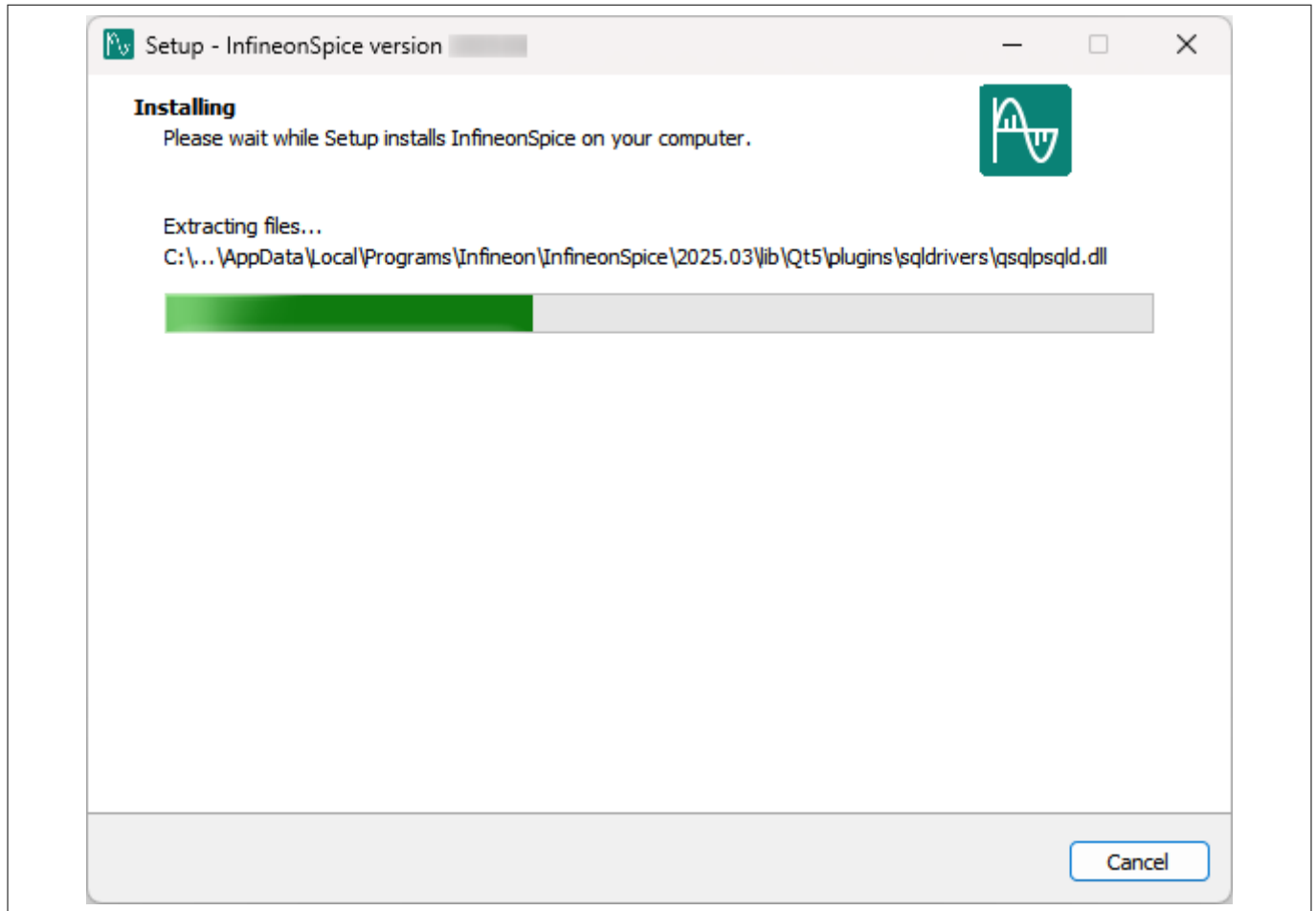


Figure 4 **Installing the software**

After the installation process has been completed you can choose to directly run InfiniteSpice (Figure 5) and display the Release Notes. If you do not want it, just un-tick the according option and click **Finish**.

1 Installing InfiniteonSpice

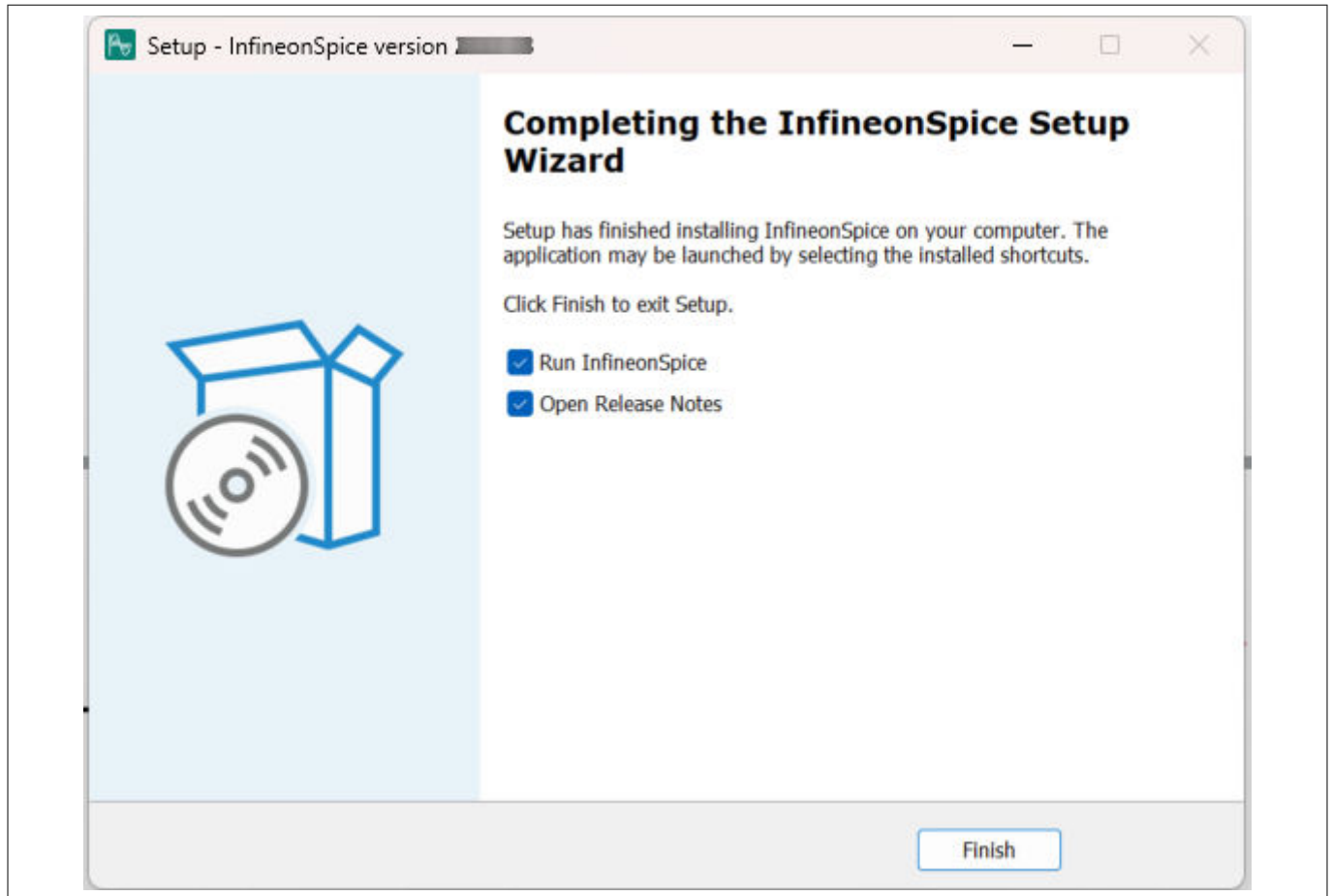


Figure 5 **Installation complete**

1.2.2 **Custom Installation**

This chapter explains the custom installation of InfiniteonSpice on your computer.

Run the InfiniteonSpice_2026.2_Windows_x64.exe installer and follow the prompts.

First of all you can select the install mode. Choose if you want to install the software only for you (recommended) or for all users.

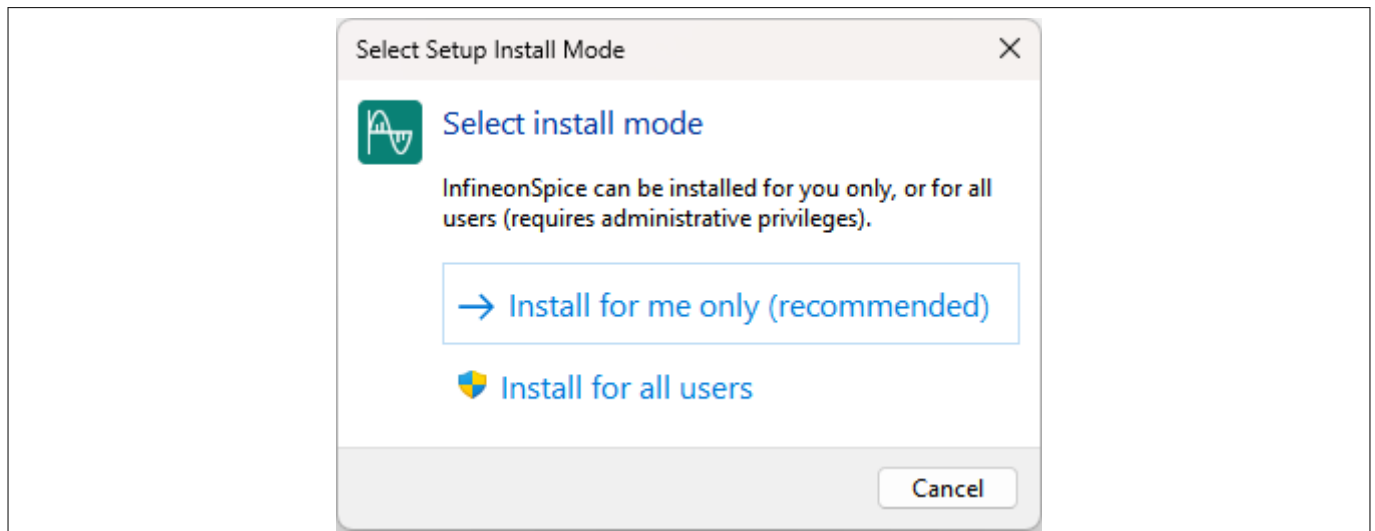


Figure 6 **Select install mode**

1 Installing InfineonSpice

Read the license agreement document presented on the next page of the installation wizard and select **I accept the agreement**. The installation cannot proceed without this. Click **Next** to proceed. Or click **Cancel** if you wish to close the installation wizard.

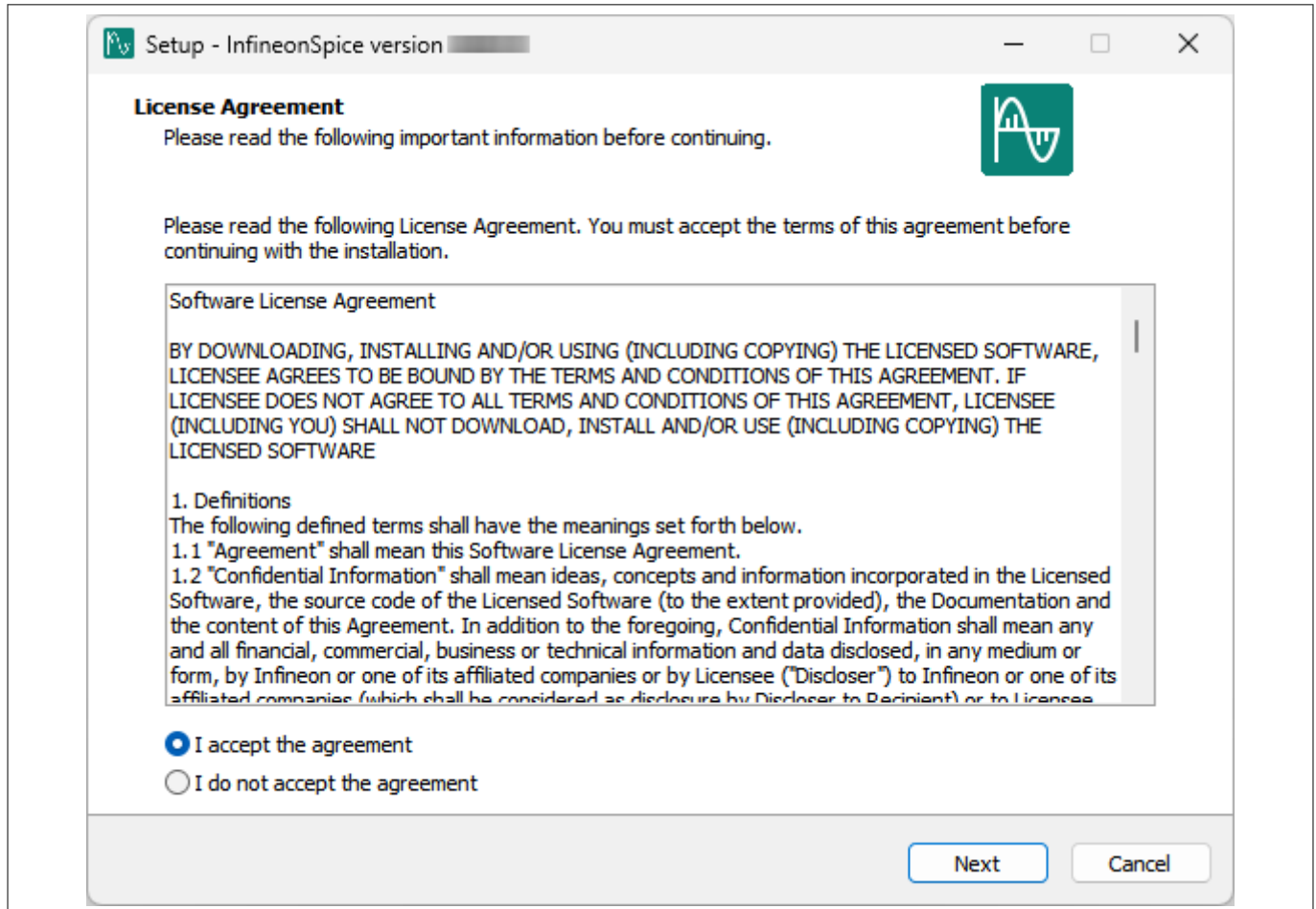


Figure 7 **License Agreement**

On the next page please choose **Custom Installation** and proceed with **Next**.

1 Installing InfiniteonSpice

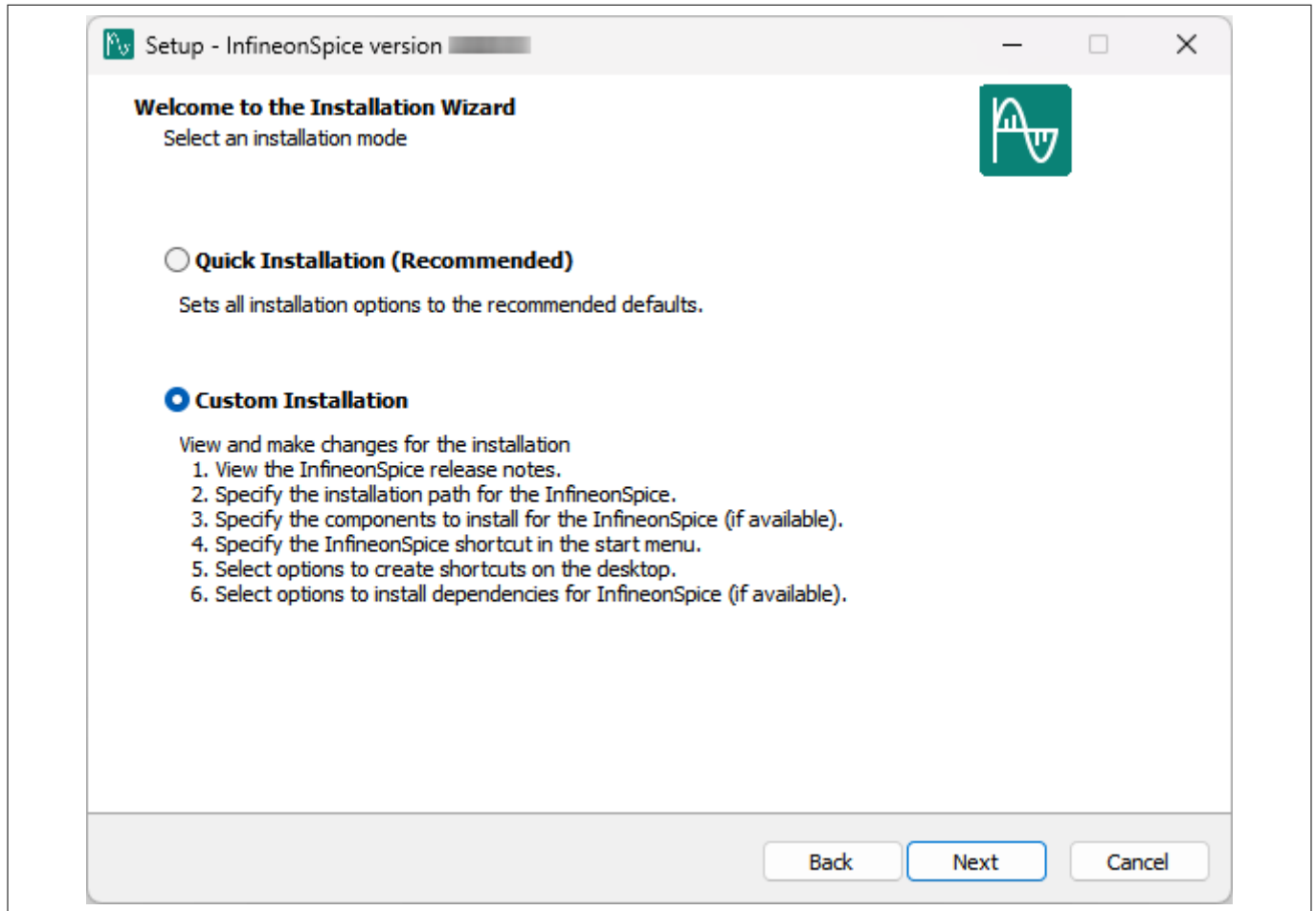


Figure 8 Custom installation

If you wish to read the Release Notes you can follow the link in the Information field.

1 Installing InfiniteSpice

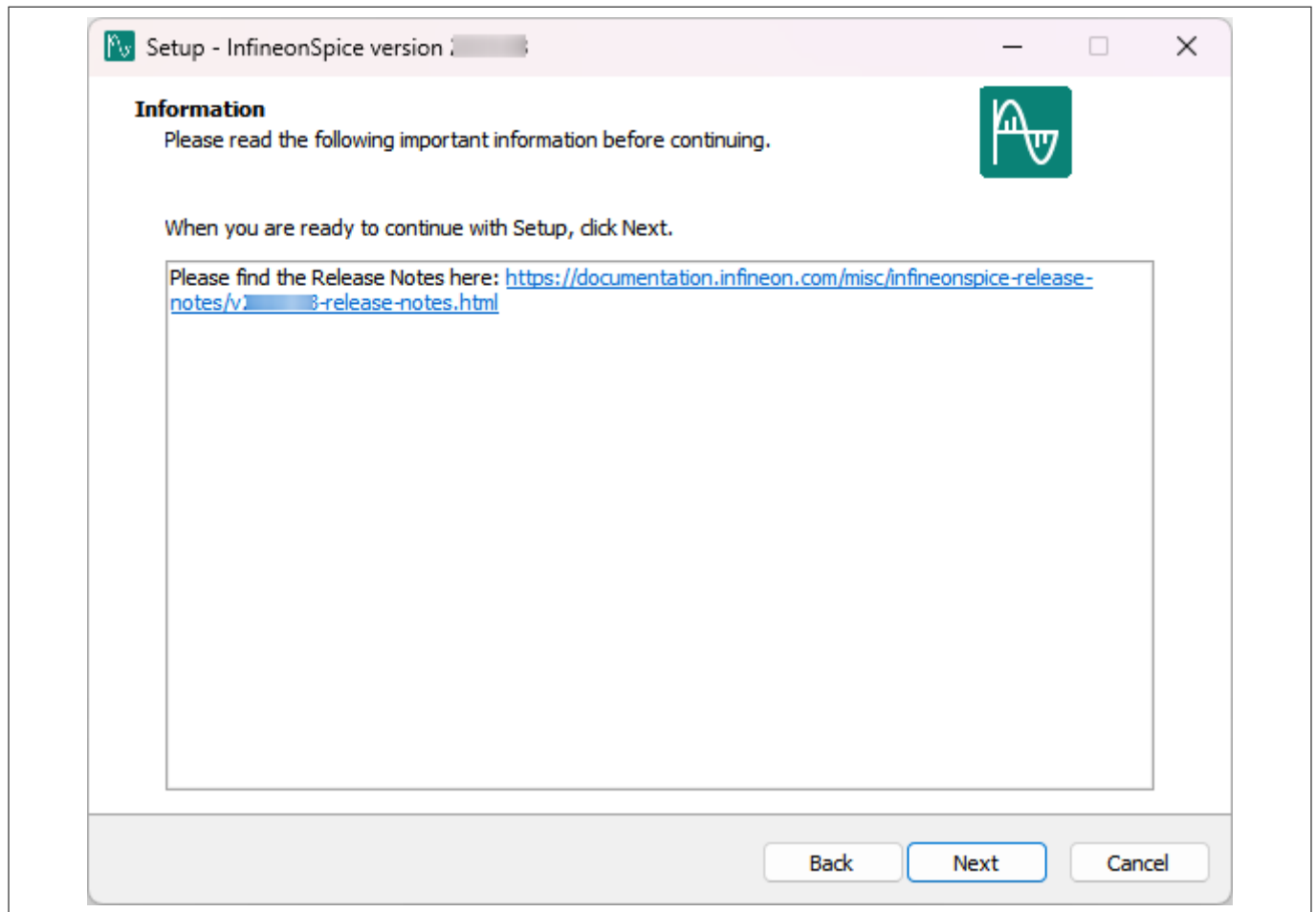


Figure 9 Information

Press **Next** and choose the location for the installation on your local disk. By default, the InfiniteSpice Software will be installed to:

C:\Users\<User>\AppData\Local\Programs\Infineon\InfiniteSpice\2026.2

1 Installing InfiniteonSpice

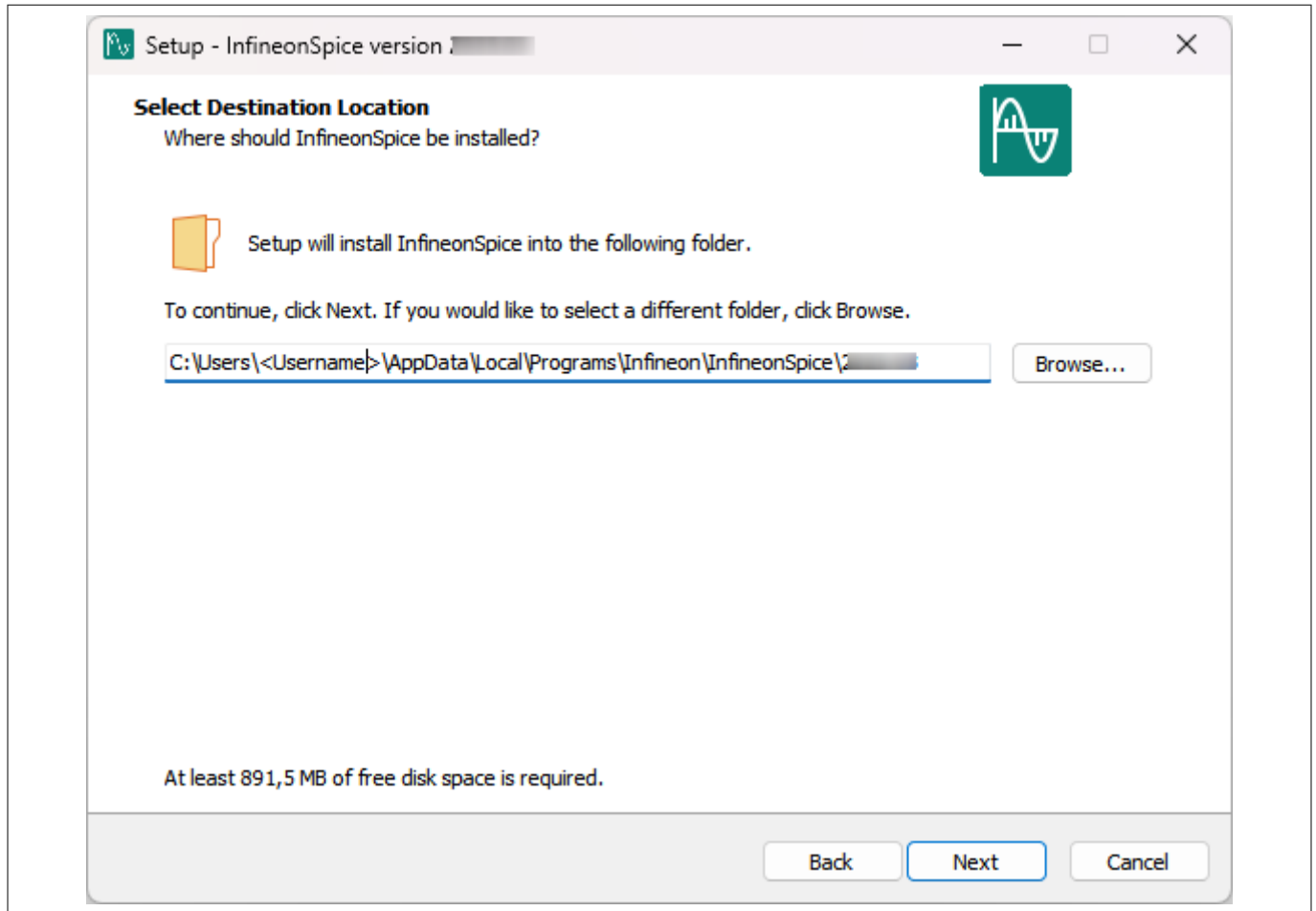


Figure 10 **Destination Location**

Note: *If you change the default path it might be necessary to install the software as administrator in order to run properly*

Note: *You can change the directory, but do not use spaces in the installation directory name. Also, do not use common illegal characters, such as: / : * ? " < > |. Refer to Naming directories and files*

Keep or enter a destination for the installation and press **Next** to proceed.
You can choose a Start Menu folder where to place the program's shortcut.

1 Installing InfiniteSpice

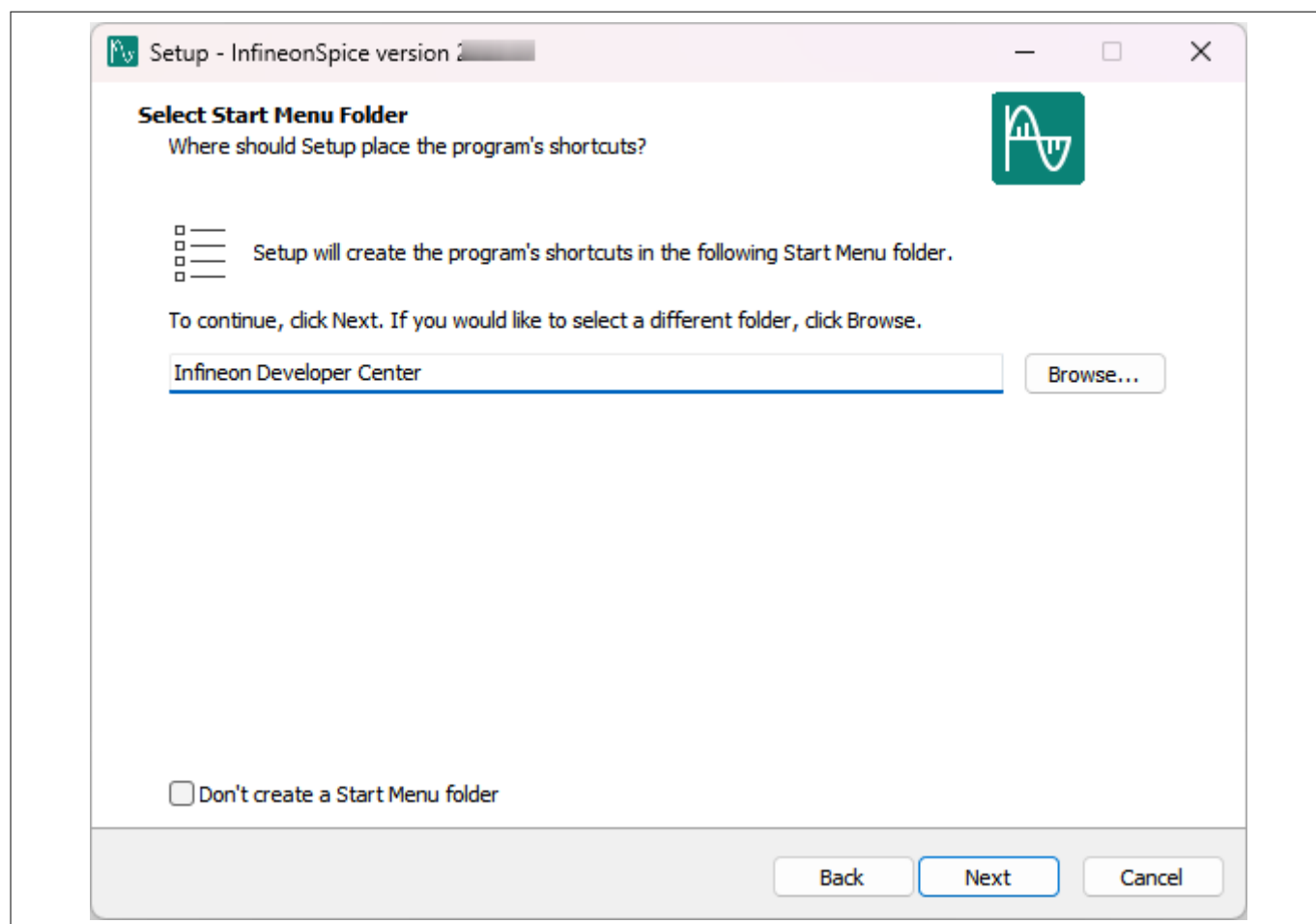


Figure 11 **Start Menu Folder**

Click **Next** to proceed or **Back** to return.

The InfiniteSpice installer can create a desktop shortcut for you and it is enabled by default. Click **Next** to proceed.

1 Installing InfiniteSpice

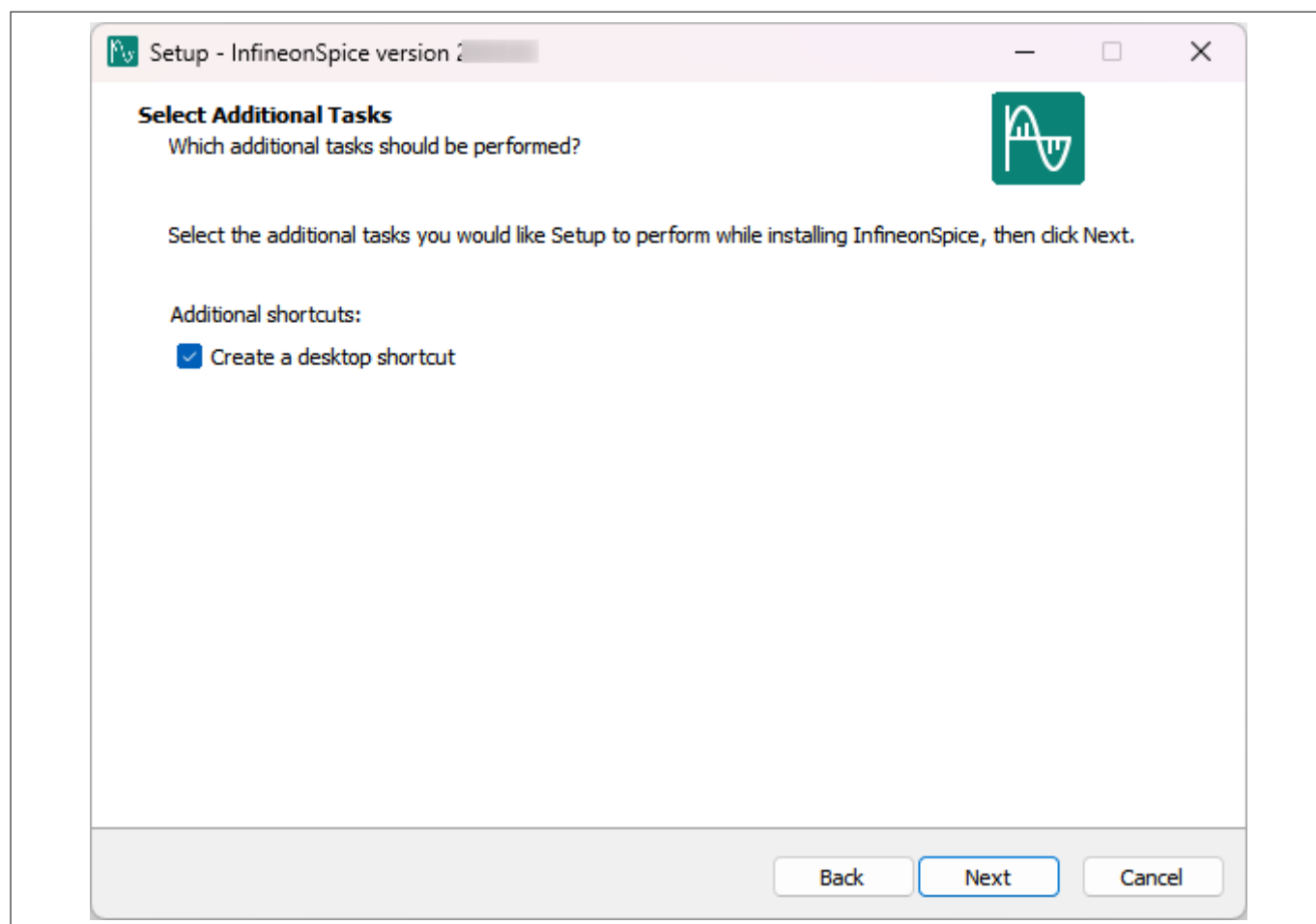


Figure 12 Create Desktop shortcut

The next page shows a summary of actions to be performed by the installer. Please verify the items before proceeding. Click **Install**.

1 Installing InfiniteSpice

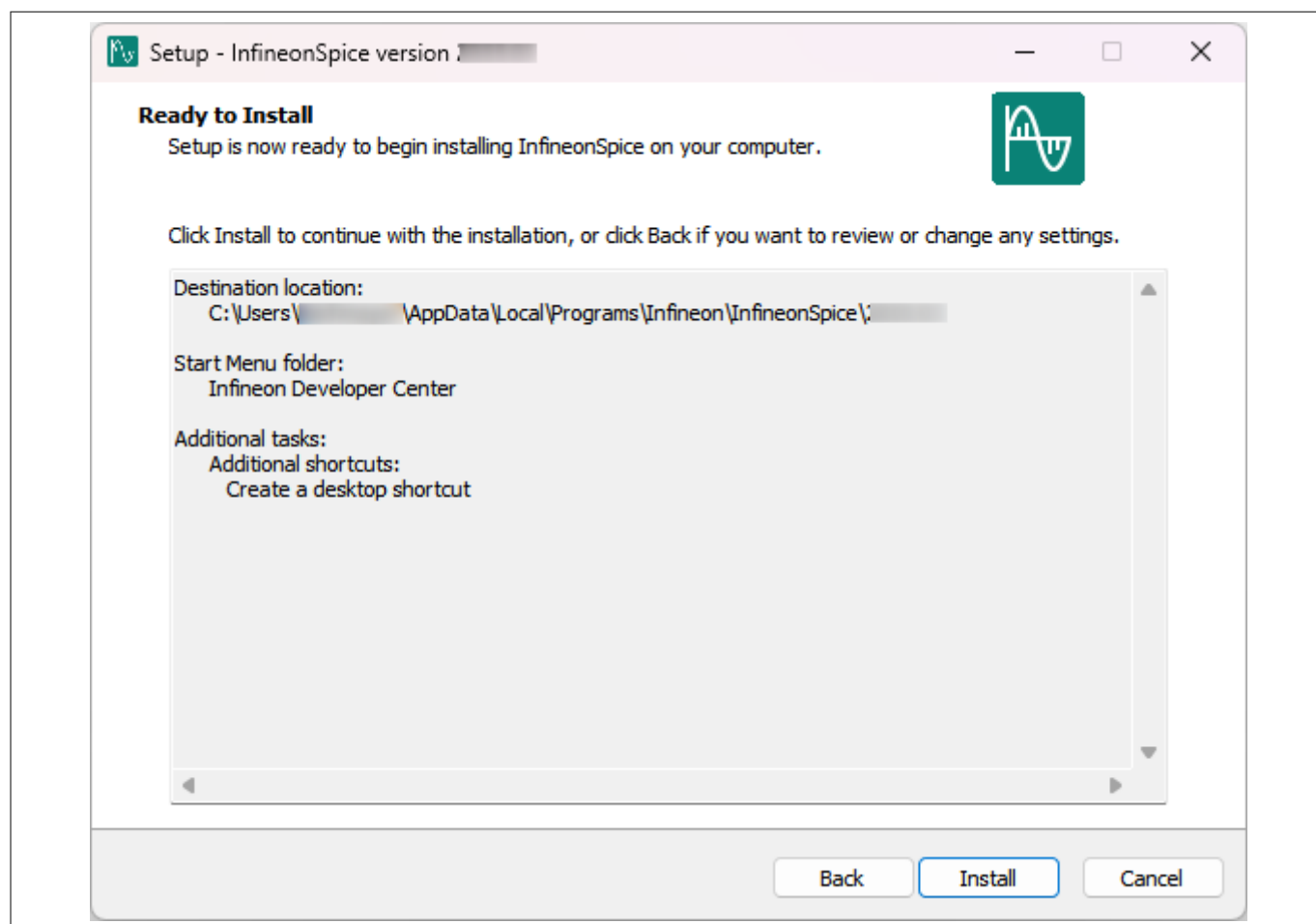


Figure 13 **Ready to install**

The following wizard will display the progress of the installation.

1 Installing InfiniteonSpice

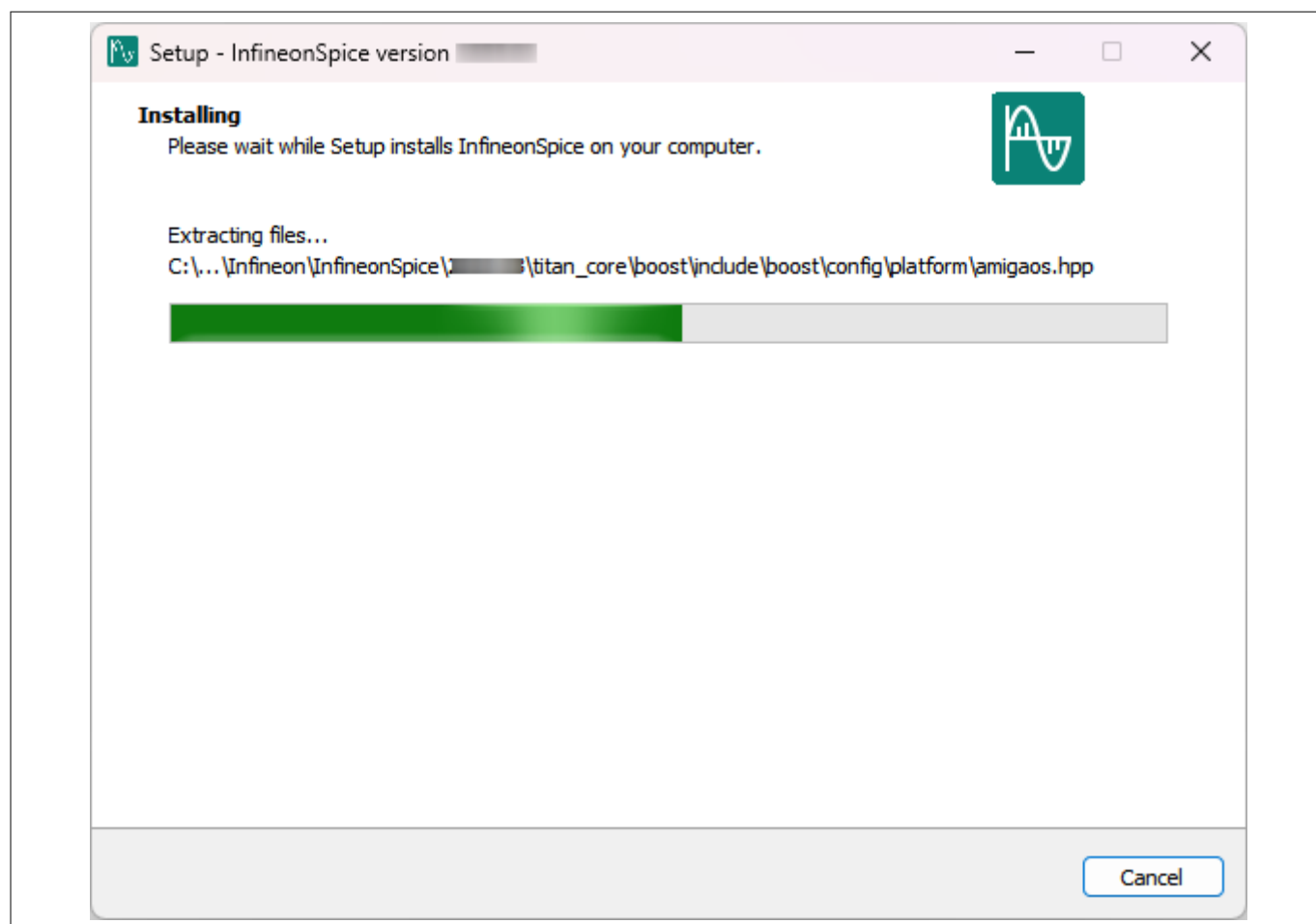


Figure 14 **Installing**

After the installation process has been completed you can choose to directly run InfiniteonSpice and display the Release Notes. If you do not want it, just un-tick the according option and click **Finish**.

1 Installing InfiniteonSpice

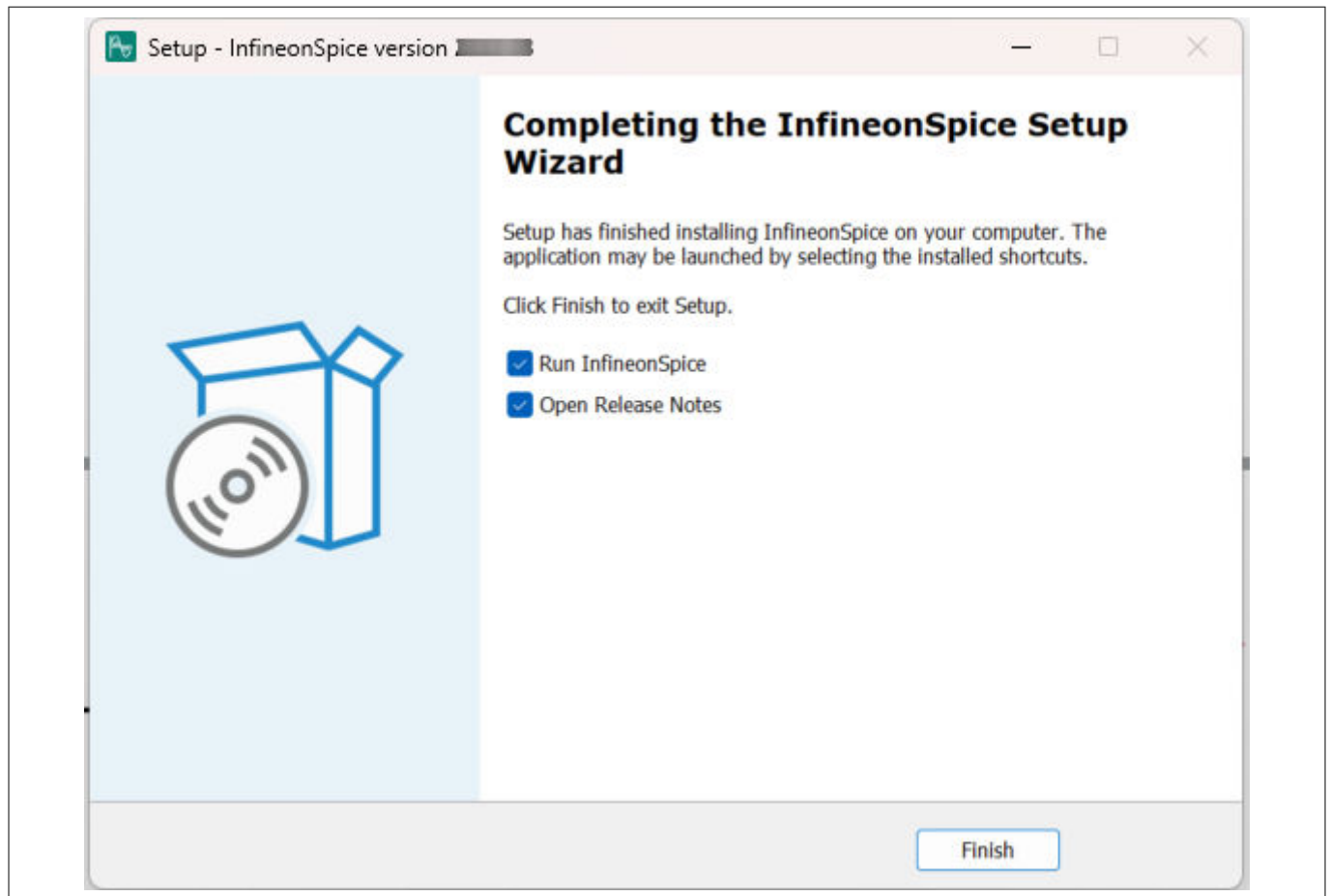


Figure 15 Installation complete

Whenever you wish to cancel the installation process in between press **Cancel** and choose **Yes** in the following prompt.

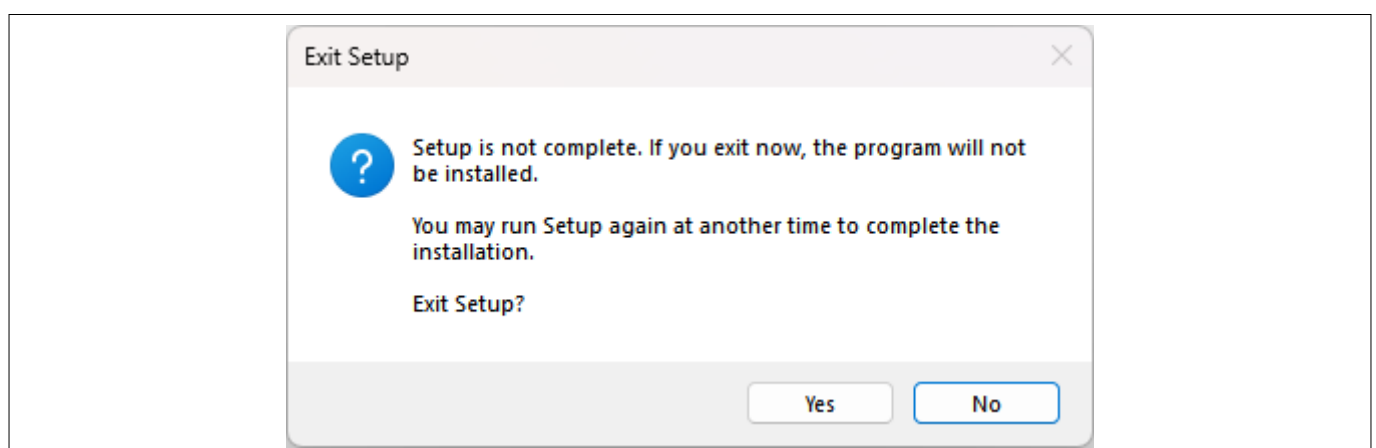


Figure 16 Cancel installation

1 Installing InfiniteonSpice

1.2.3 Uninstall

If you wish to remove the software from your computer please do as follows.

To uninstall the software search for InfiniteonSpice in your Windows Start Menu and choose "Uninstall". See image below (Figure 17).

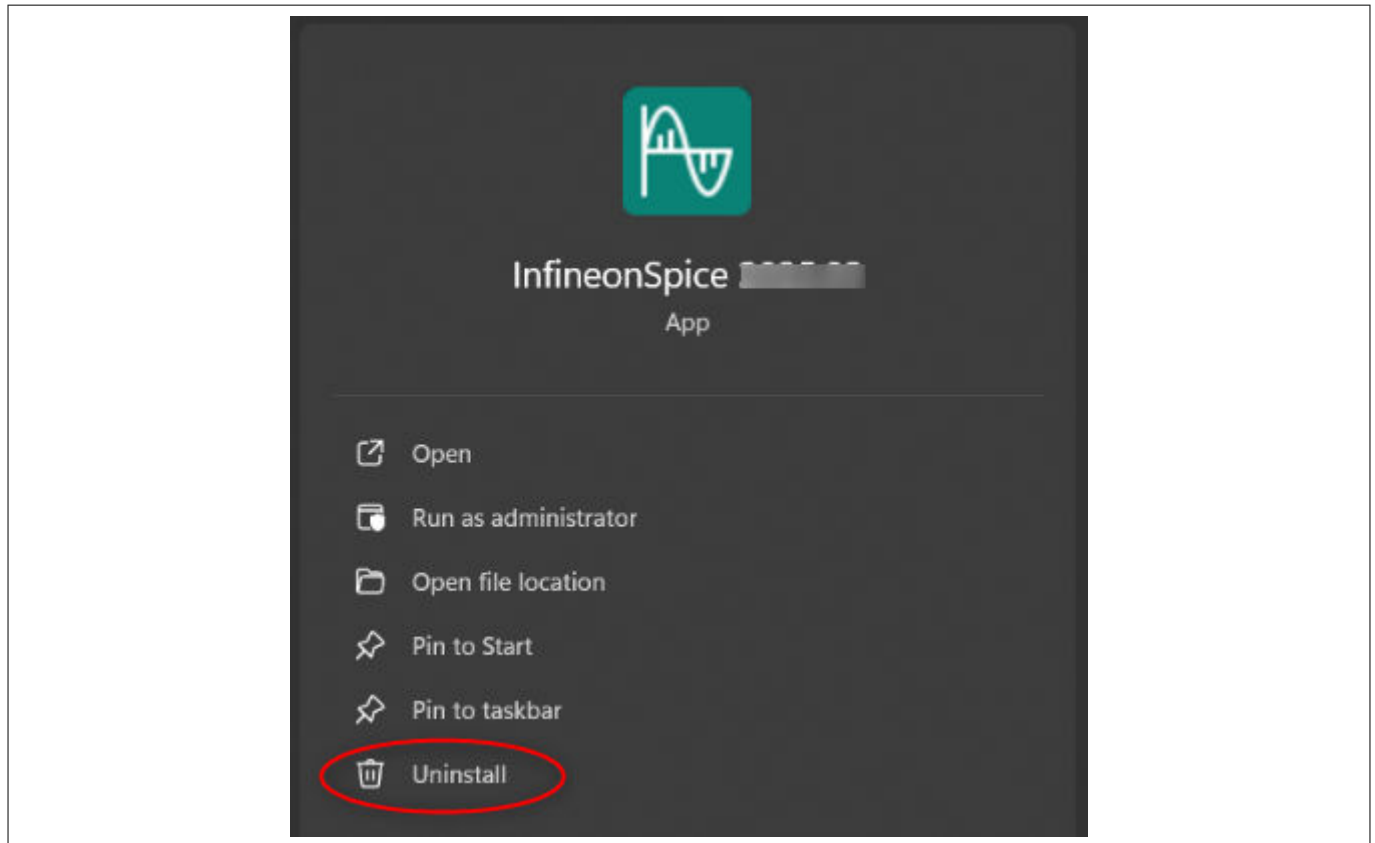


Figure 17 Uninstall the software

A separate window will be opened. Right-click on the version you wish to remove and select **Uninstall**.



Figure 18 List of installed apps

Note: After the InfiniteonSpice software has been removed, it might not be possible anymore to edit files created with the application.

2 Getting started

This getting started guide takes you through all steps required for designing a schematic and simulating its operation.

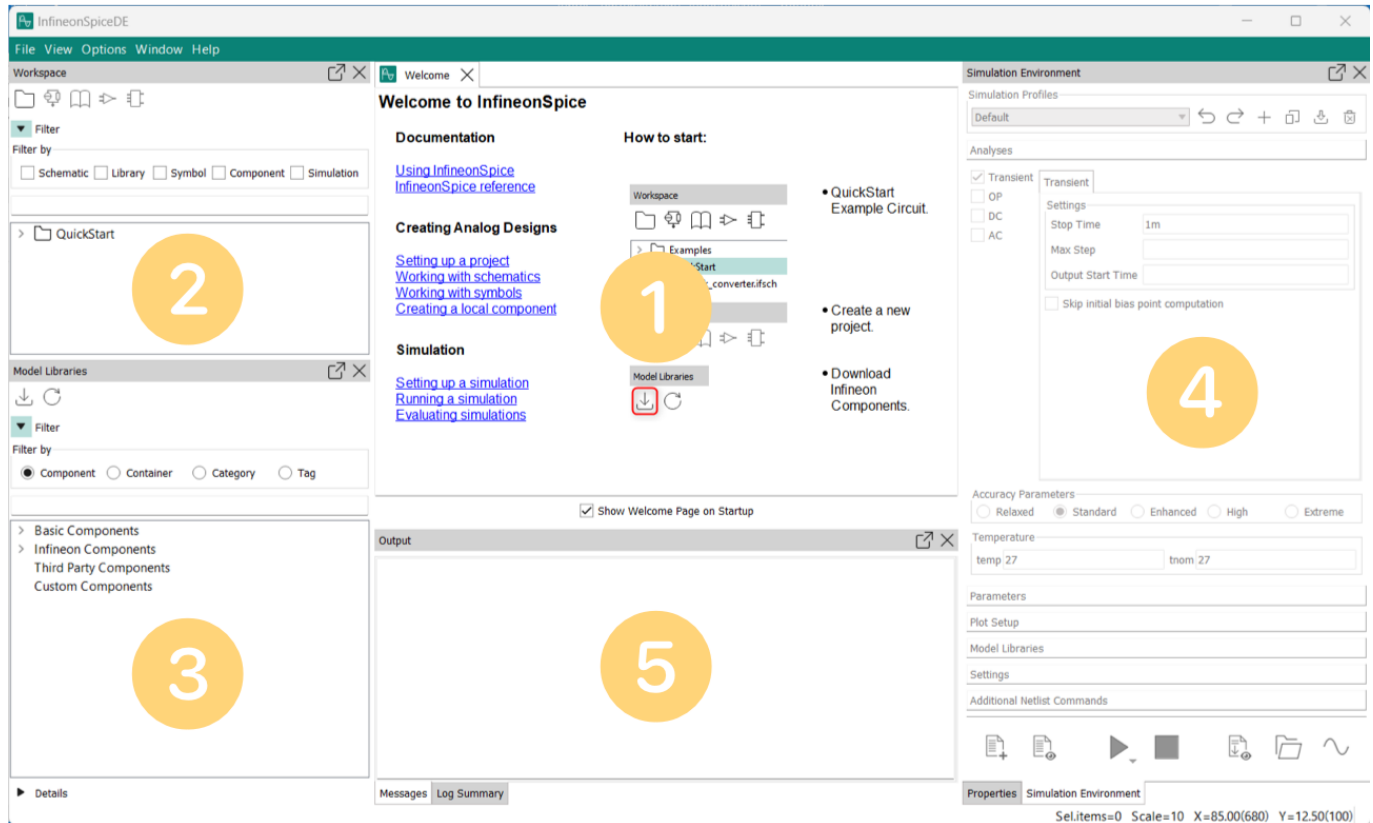
It includes:

- Creating a new project
- Creating a new schematic
- Creating a simulation profile
- Running a simulation
- Viewing the simulation results

2 Getting started

2.1 Introduction

This getting started guide shows how to work with the InfiniteSpice design environment and waveform viewer. When you have installed InfiniteSpice and start it for the first time, you are presented with the design environment:



The main areas of the window are:

1. Work area, in which you can display and edit schematics, symbols, and other documents. If no document is open, it displays helpful links and tips.
2. Workspace, which organizes your projects and their elements.
3. Model libraries, which contain components that you can use to construct schematics.
4. Simulation setup, in which you can configure and run simulations.
5. Output window, which displays messages and logging information.

In a typical workflow, you create the schematics in the workspace and edit them in the work area. If you need any custom components, you also create them in the workspace and edit them in the work area.

When the schematic is ready, you set up the parameters for the simulation and run the simulation. The results of simulations are displayed and analyzed in a separate window, which is not shown above.

The elements of the main window are described in detail in [Windows and views](#).

The rest of this guide walks you through this workflow, using a simple schematic as an example.

2.2 Creating a project

Anything related to a schematic you want to simulate is grouped in a project in the workspace, so you need to create the project first.

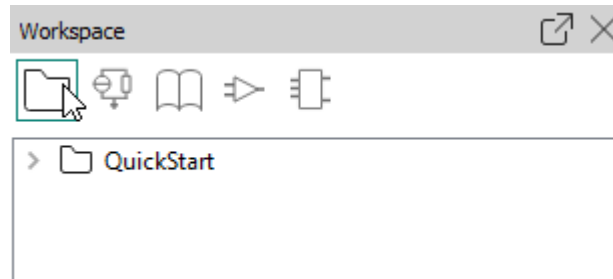
Before you begin

InfiniteSpice must be running to create a project. If it does not run, launch it now. You should be seeing the design environment shown in [Introduction](#).

2 Getting started

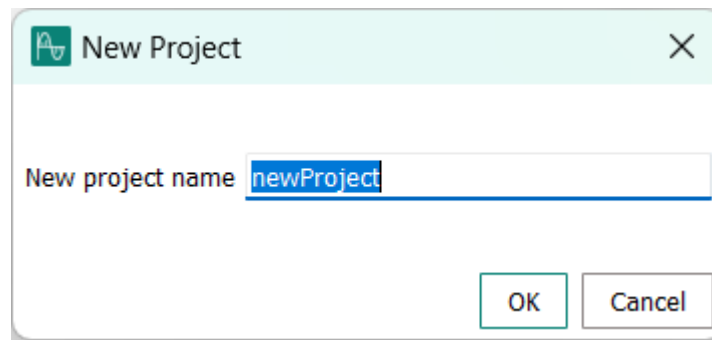
Steps

1. Click the  button in the workspace.



You can also choose **File > New > New Project**.

A dialog in which you can name the new project opens.



2. Name the new project “Tutorial” and click **OK**.

Results

The new project appears in the **Workspace** view.

It already contains an empty schematic with the same title as the project (Tutorial.ifsch, in this case).

The schematic is automatically displayed in the work area. It contains no components, but a grid of dots indicates the background.

What to do next

Design a circuit as described in [Creating a schematic](#).

2.3 Creating a schematic

Schematics are created in the **Workspace** view and populated in the work area.

Before you begin

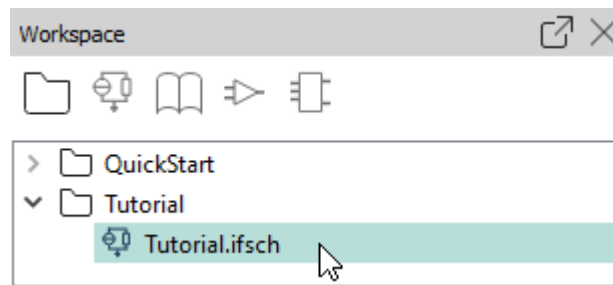
The tutorial project must have been created, as described in [Creating a project](#). It must contain an empty schematic.

In this example, you will create a simple low-pass filter.

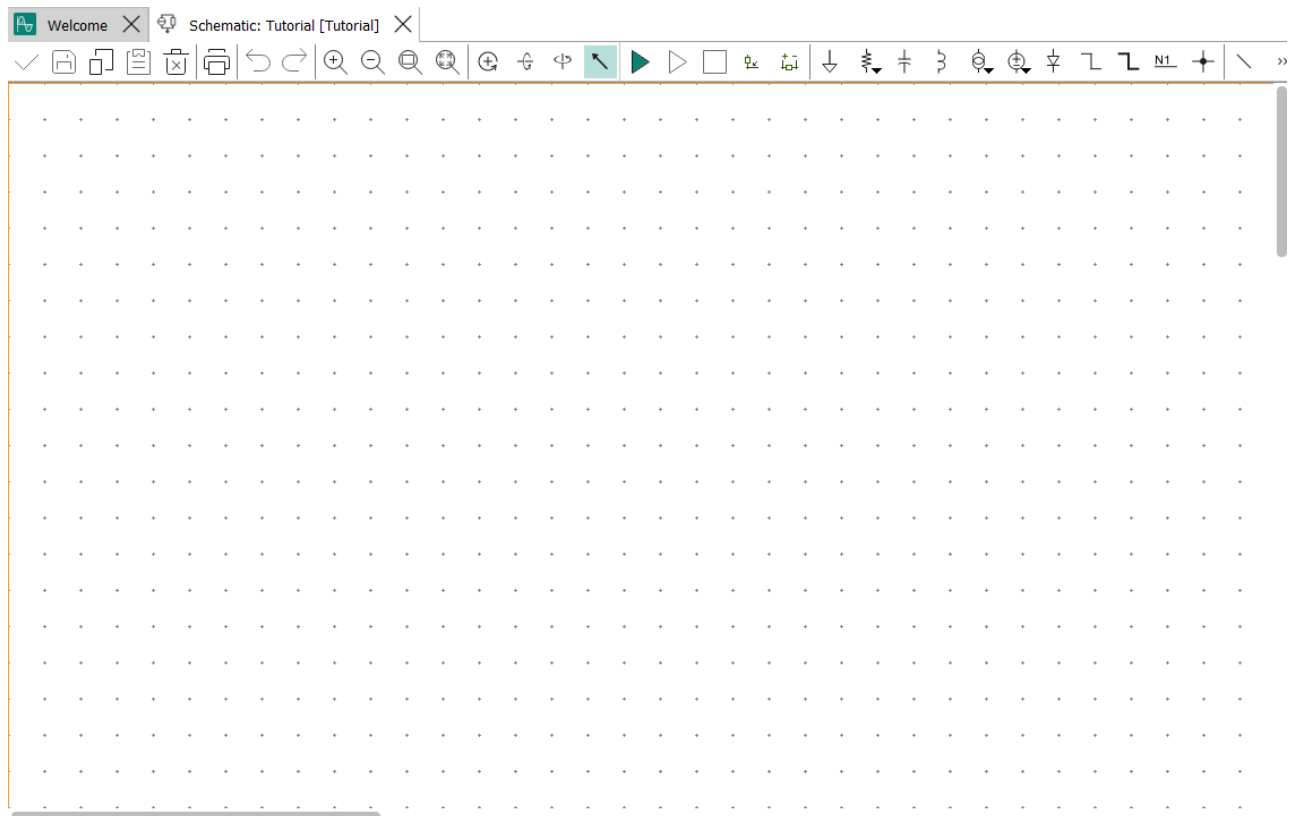
Steps

1. If the empty schematic is not already open, open it by double-clicking its file in the **Workspace** view.

2 Getting started



The schematic is displayed in the work area.



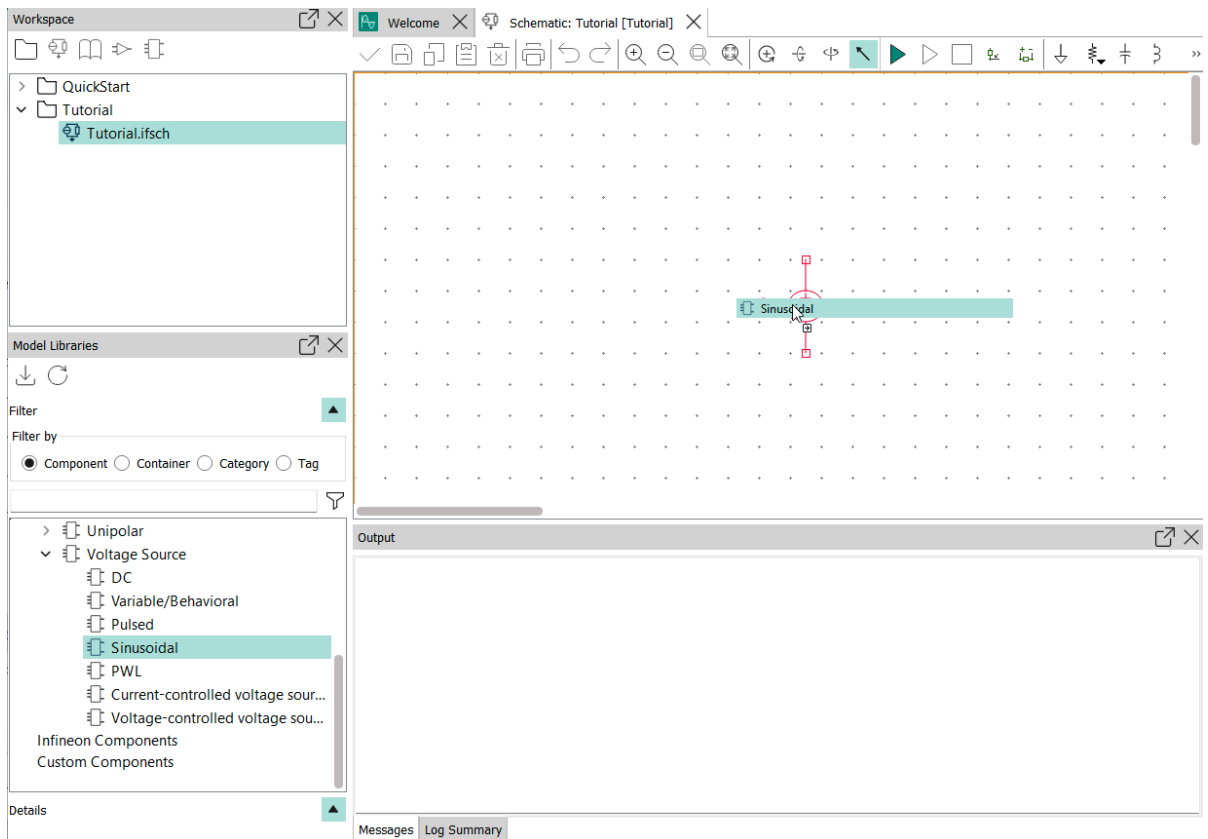
2. Place the voltage source.

We are using a sinusoidal voltage source, which is a basic component.

The following steps show how to place such components from the model libraries, which works in the same way for more complicated components. Basic components can also be placed using the toolbar icons in a schematic view. Except for how the desired component is chosen, the procedure is the same as described here.

- a. In the **Model Libraries** view, open the Basic Components section by clicking the triangle in front of its name, and then open Voltage Source.
- b. Drag the entry **Sinusoidal** from the library to the work area.

2 Getting started



Instead of dragging the component, you can also double-click it in the library.

The cursor changes its form  to indicate that you can now place sinusoidal voltage sources.

- c. Click the mouse in the location where you want to put the voltage source.
- d. Press Esc to return to the standard cursor.

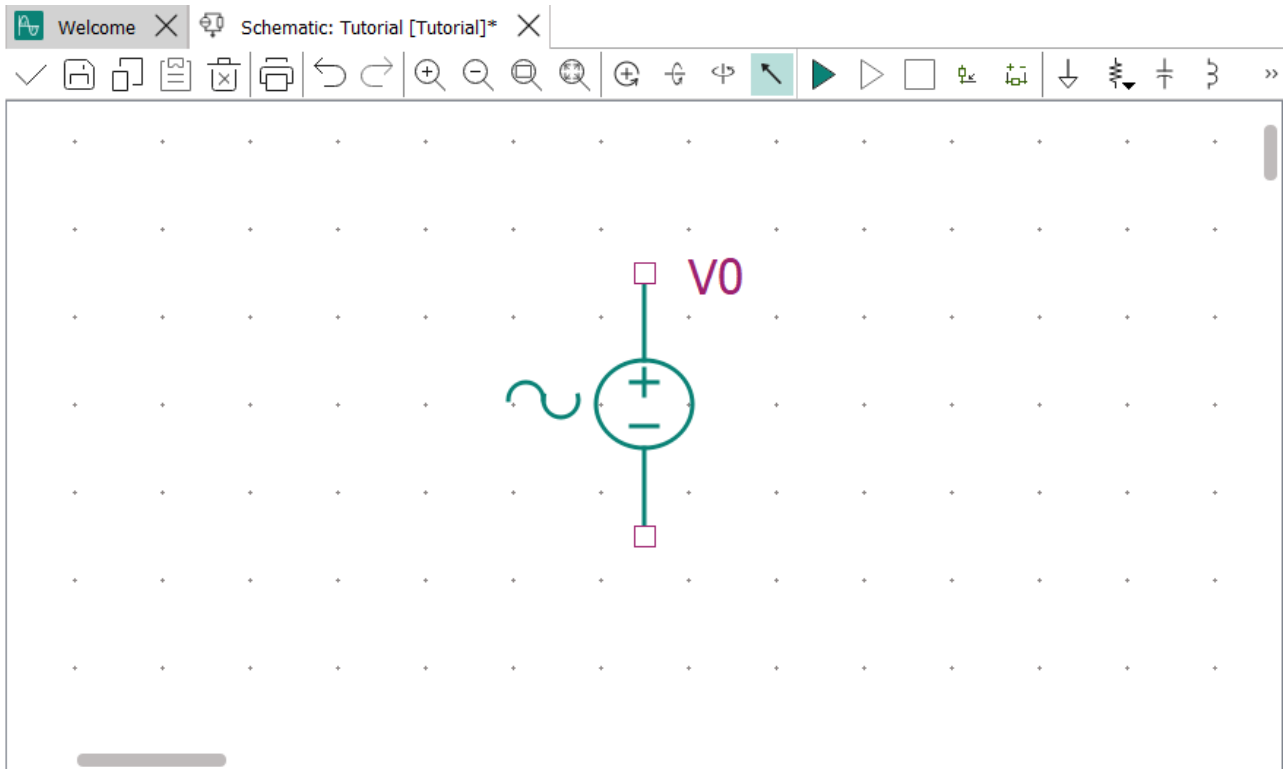
If you do not press Esc or choose a different component, you can continue clicking to place additional voltage sources.

The schematic now contains a sinusoidal voltage source.

3. Zoom in to get a better view of the component.

To do so, click the Zoom In  tool until the voltage source symbol is large enough or click the Zoom Region  tool and drag a conveniently sized rectangle around the area on which you want to zoom in. You should now be able to clearly see the symbol in the schematic.

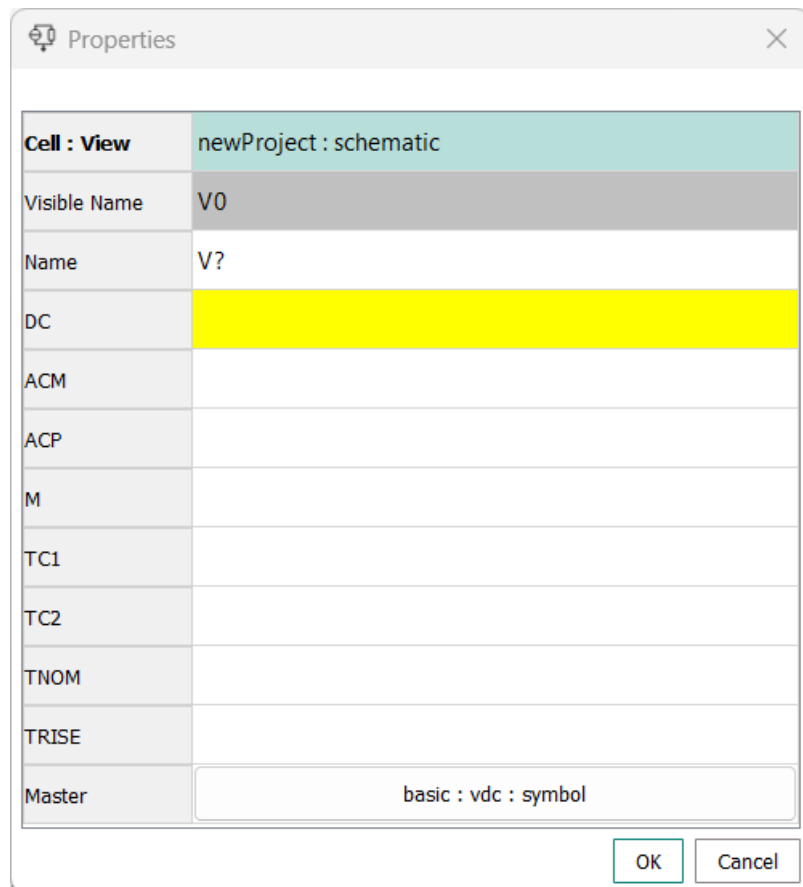
2 Getting started



The dots in the background indicate the grid that can assist you in correctly placing and aligning components.

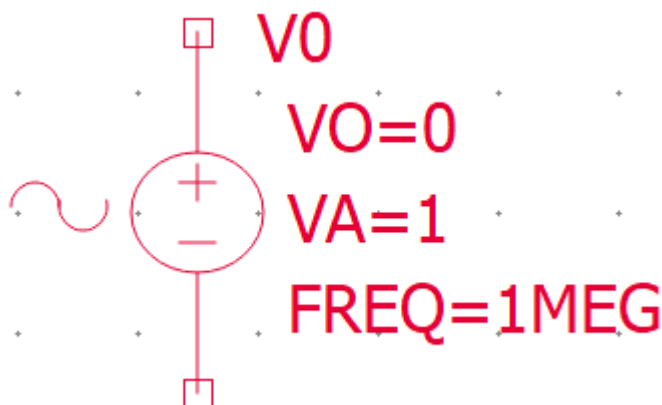
4. Edit the properties of the voltage source to fit the schematics' requirements.
 - a. Choose the Select Mode tool  and double-click the voltage source.
Alternatively, you can also select the voltage source and press Q or right-click it and choose **Properties**.
The **Properties** dialog opens.

2 Getting started



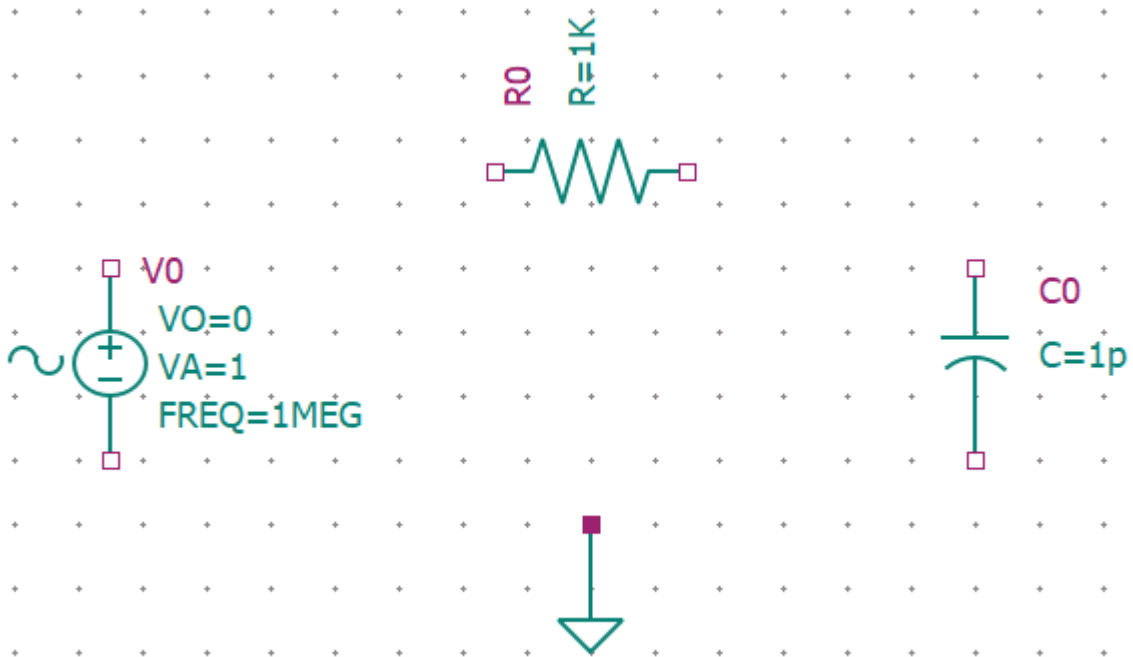
The dialog lists all available parameters of the selected component. Parameters that are mandatory are highlighted in yellow. Brief explanations of the parameters are available by hovering the mouse over the parameter name.

- b. Enter the following values: VO 0, VA 1, FREQ 1MEG, TD 0. Click **OK**.
The most important of the specified values are displayed beside the voltage source.



5. In the same manner as the voltage source, drag a capacitor, a ground, and a resistor to the schematic. Do not modify these components' parameters.
6. Select the resistor, right-click it, and choose Rotate from the context menu.
Your schematic now looks similar to this:

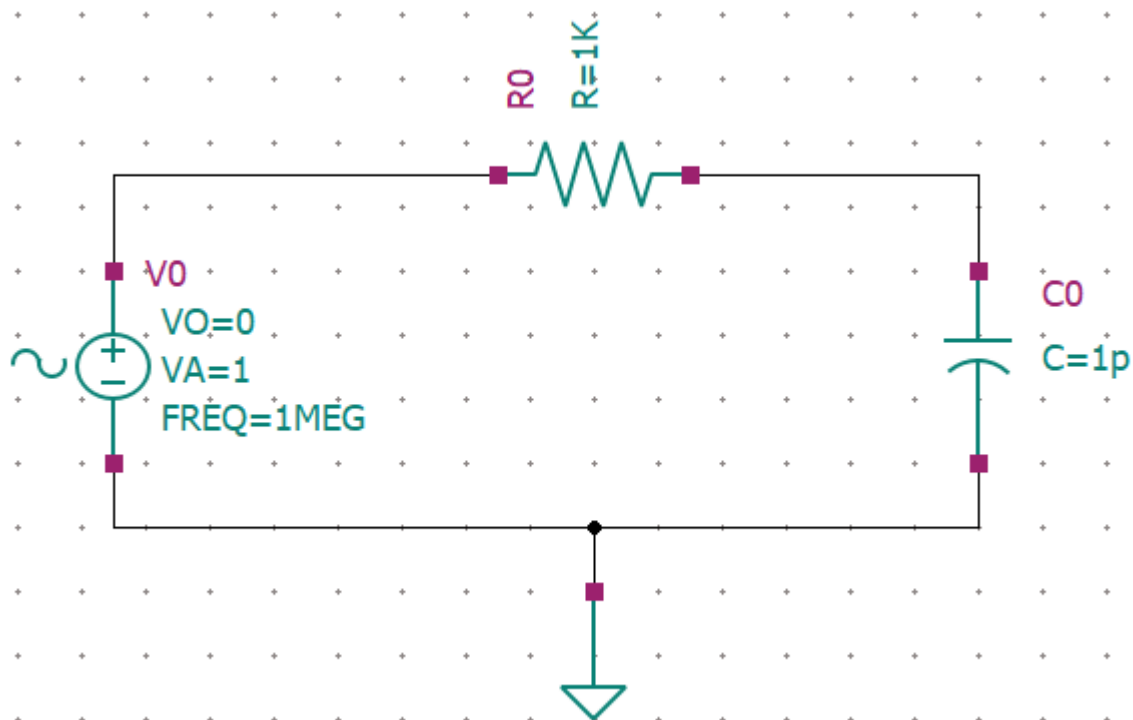
2 Getting started



7. Connect the components with wires.

- Choose the wire tool .
- Click the upper pin of the voltage source and then click the left pin of the resistor. Both pins are connected by a wire-
- In the same way, connect the resistor to the capacitor and the ground to the capacitor and the voltage source.

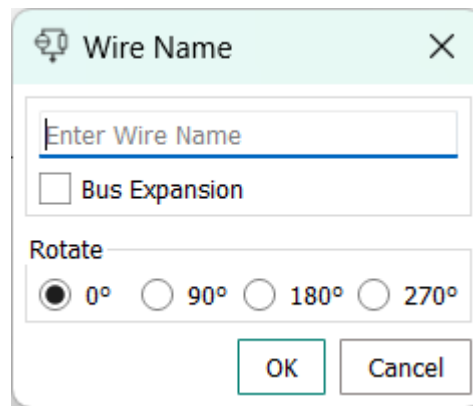
The schematic now looks like this:



8. Label the wires to the resistor.

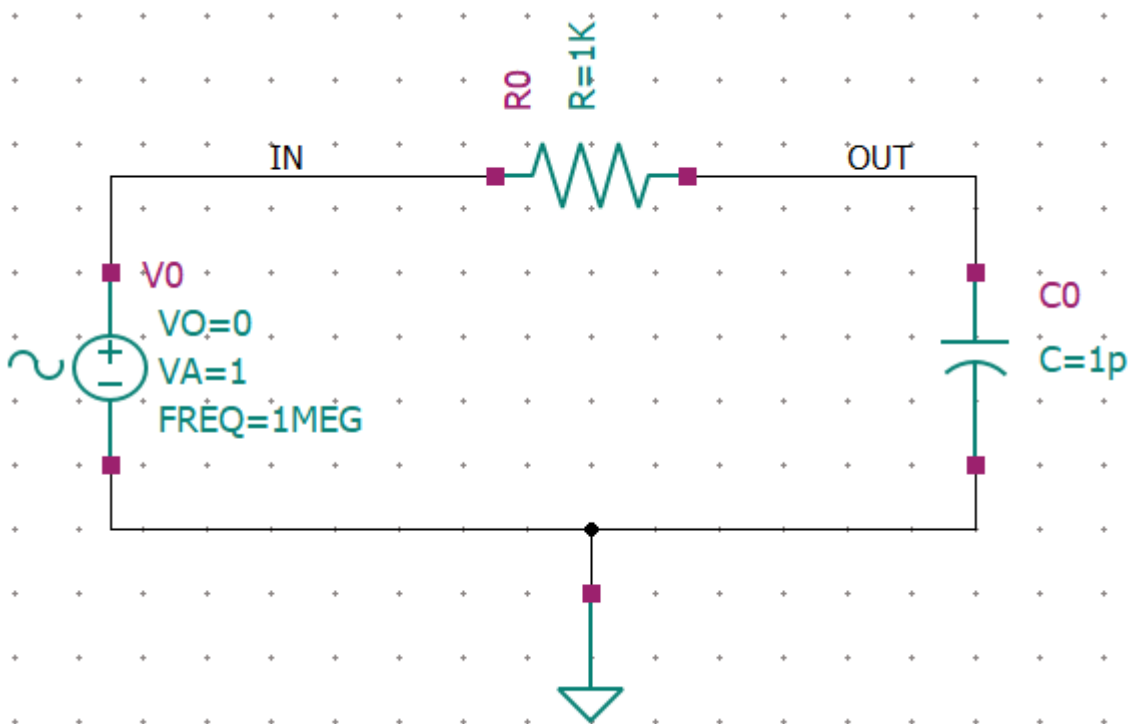
- Choose the wire labeling tool . The **Wire Name** dialog opens:

2 Getting started



- b. Enter IN as the wire name and click **OK**.
- c. Click the connection between the resistor and the voltage source.
The wire is labeled "IN".
- d. Choose N1 again, enter OUT as the wire name, and click the wire from the resistor to the capacitor.
The wire is labeled "OUT".

The wires are now labeled:



These labels can be referenced when setting up the simulation.

9. Press Ctrl+S to save the schematic.

Results

You have created a schematic for a simple low-pass filter the operation of which can be simulated in InfineonSpice.

What to do next

Set up the simulation as described in [Setting up a simulation](#).

2 Getting started

2.4 Setting up a simulation

The parameters with which a simulation is performed are configured in the **Simulation Environment** view. Setting up a simulation with the appropriate parameters ensures that it will produce meaningful results.

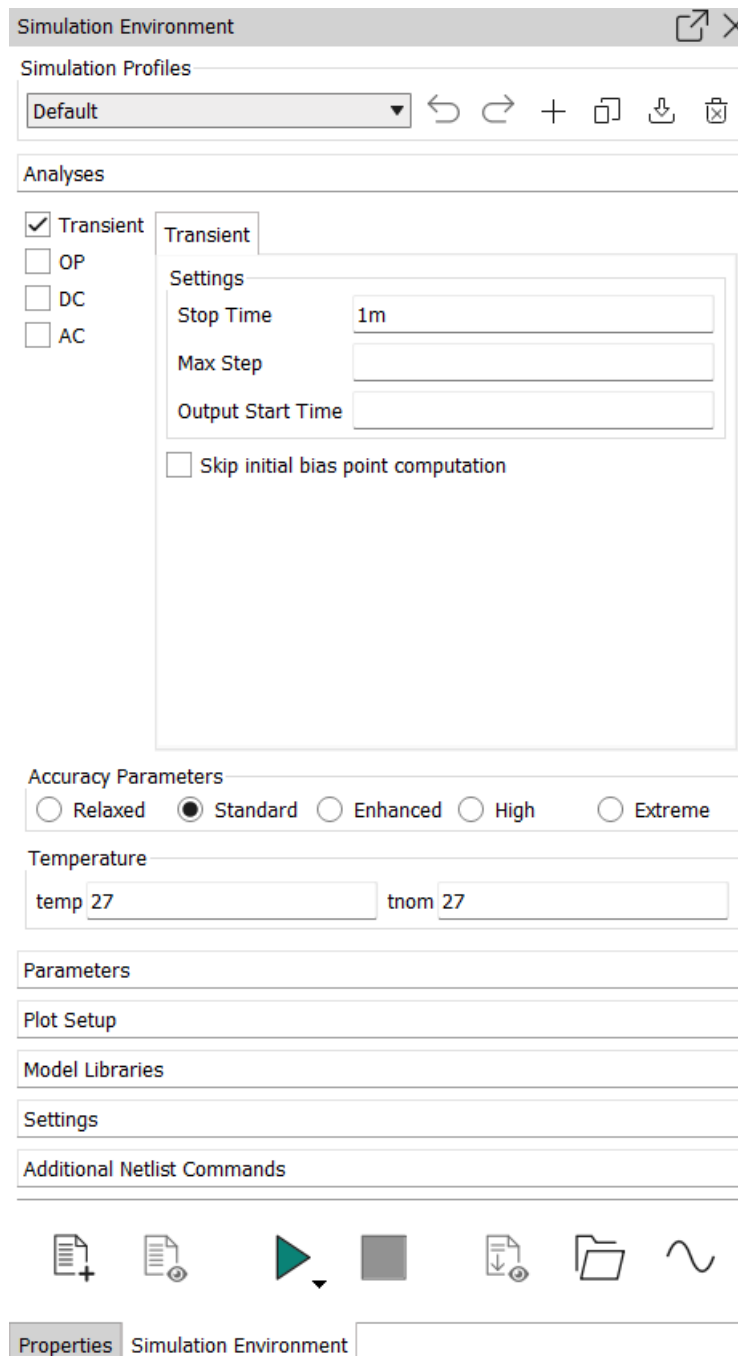
Before you begin

You need a schematic that you can simulate. We recommend using the schematic that you have created in [Creating a schematic](#). If you use a different schematic, the process is the same, but some of the settings may not be available or may need to be different.

Steps

1. If the schematic that you want to simulate is not already open, open it.
For example, you can double-click it in the **Workspace** view.
2. If the **Simulation Environment** view is not open, choose **View > Simulation Environment** to open it.

2 Getting started



3. Create a new simulation profile.

Simulation profiles collect the settings with which a simulation is run. By storing settings in a profile, you can quickly recall them for future simulations in the same project or in others.

- a. In the toolbar of the **Simulation Environment** view, click .
- The toolbar is overlaid with a text entry field.

- b. Enter TutorialProfile into the field and click **OK**.

The **Simulation Environment** view displays the new, empty profile TutorialProfile. Any changes you now make to the settings in the view change this profile but not others.

4. Specify what kind of analysis the simulation should provide.

In this case, it will perform a transient analysis.

- a. In the **Analyses** section, check the **Transient** option.

2 Getting started

If the option is already checked, leave it on. If **OP** or **DC** are checked, uncheck them.

- b. In the **Stop Time** field, enter 1u.

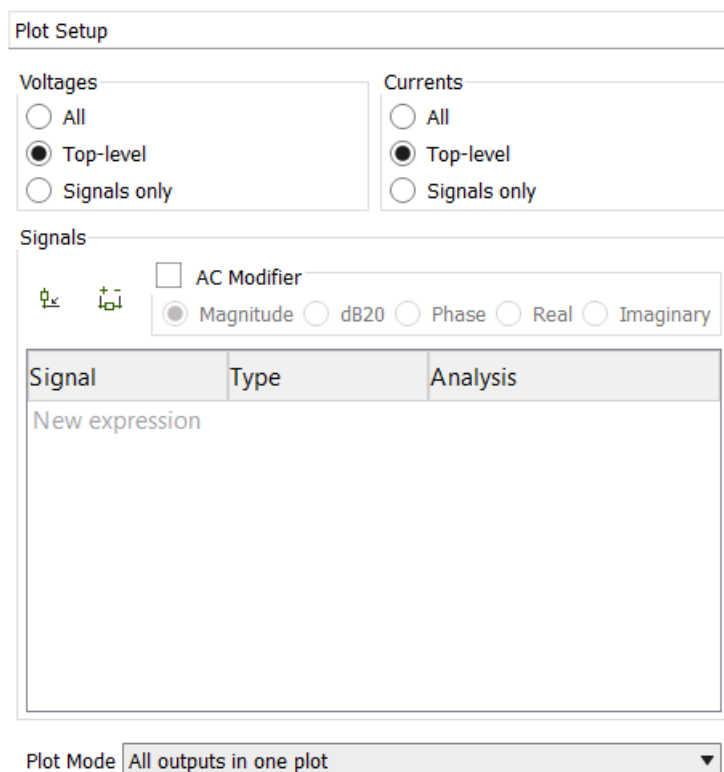
This value corresponds to one period of the sinusoidal input voltage source.

5. Specify the desired outputs to be measured.

Outputs are the signals that are plotted in the Waveform Viewer after the simulation. You can add more outputs after running the simulation, but specifying them upfront helps you see the important results immediately.

For this example, the output voltage and the current through the capacitor of the low-pass filter will be plotted.

- a. Click the **Plot Setup** section of the **Simulation Environment** view.
The section expands.



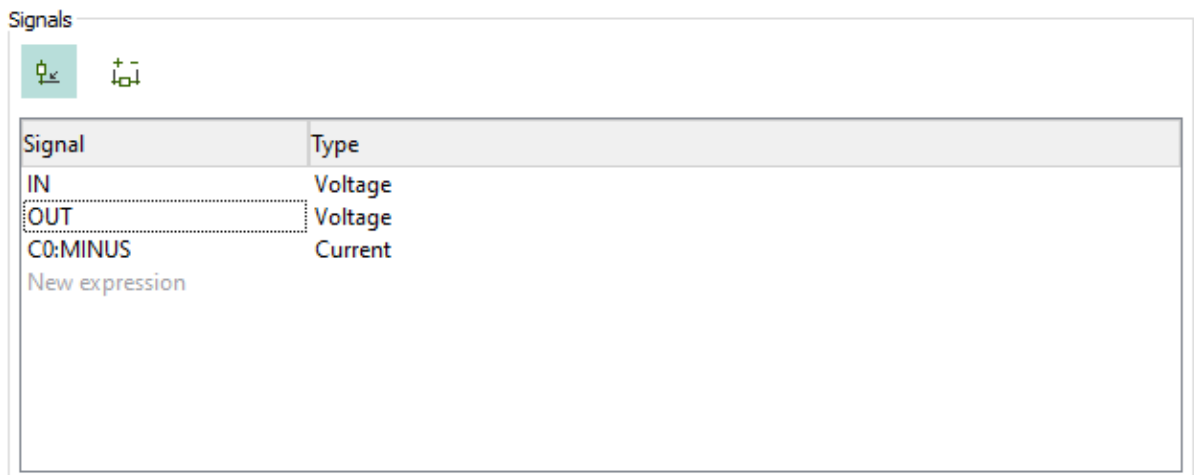
The **Plot Setup** dialog box is shown with the following settings:

- Voltages**: ☐ All, ☒ Top-level, ☐ Signals only
- Currents**: ☐ All, ☒ Top-level, ☐ Signals only
- Signals**:
 - ☐ AC Modifier
 - ☒ Magnitude, ☐ dB20, ☐ Phase, ☐ Real, ☐ Imaginary
- Signal List Table**:

Signal	Type	Analysis
New expression		
- Plot Mode**: All outputs in one plot

- b. In both the **Voltages** and **Currents** sections, choose **Signals only**.
This setting determines which information is stored in the results database. **Signals only** means that only the outputs you specify are stored. With this setting, you cannot plot additional outputs without rerunning the simulation but the simulation runs faster and uses less disk space.
- c. In the **Signals** section, choose the Select Output on Schematic tool (.
- d. Click the wire labeled "IN" between the voltage source and the resistor.
A new signal named "IN" with the type "Voltage" is added to the signal list.
- e. With the same tool, click the "OUT" wire between the resistor and the capacitor and then click the pin of the capacitor that is connected to ground.
Typically, you click a wire to add a voltage and a pin to add a current.
Two more signals are added, "OUT" (a voltage) and "C0:MINUS" (a current).

2 Getting started



f. Click the  button in the **Signals** section again to deselect the tool.

6. Choose **File > Save** to save the settings.

Results

The simulation is now set up and ready to run.

What to do next

Run the simulation as described in [Running a simulation](#).

2.5 Running a simulation

When a simulation has been set up, running it determines the schematic's electrical properties and displays them in graphs.

Running a simulation lets you determine how the schematic behaves over time under the configured conditions.

Before you begin

You must have set up a simulation for a schematic, as described in [Setting up a simulation](#). If you use a different schematic than described there or other settings, the process is the same but the results will be different.

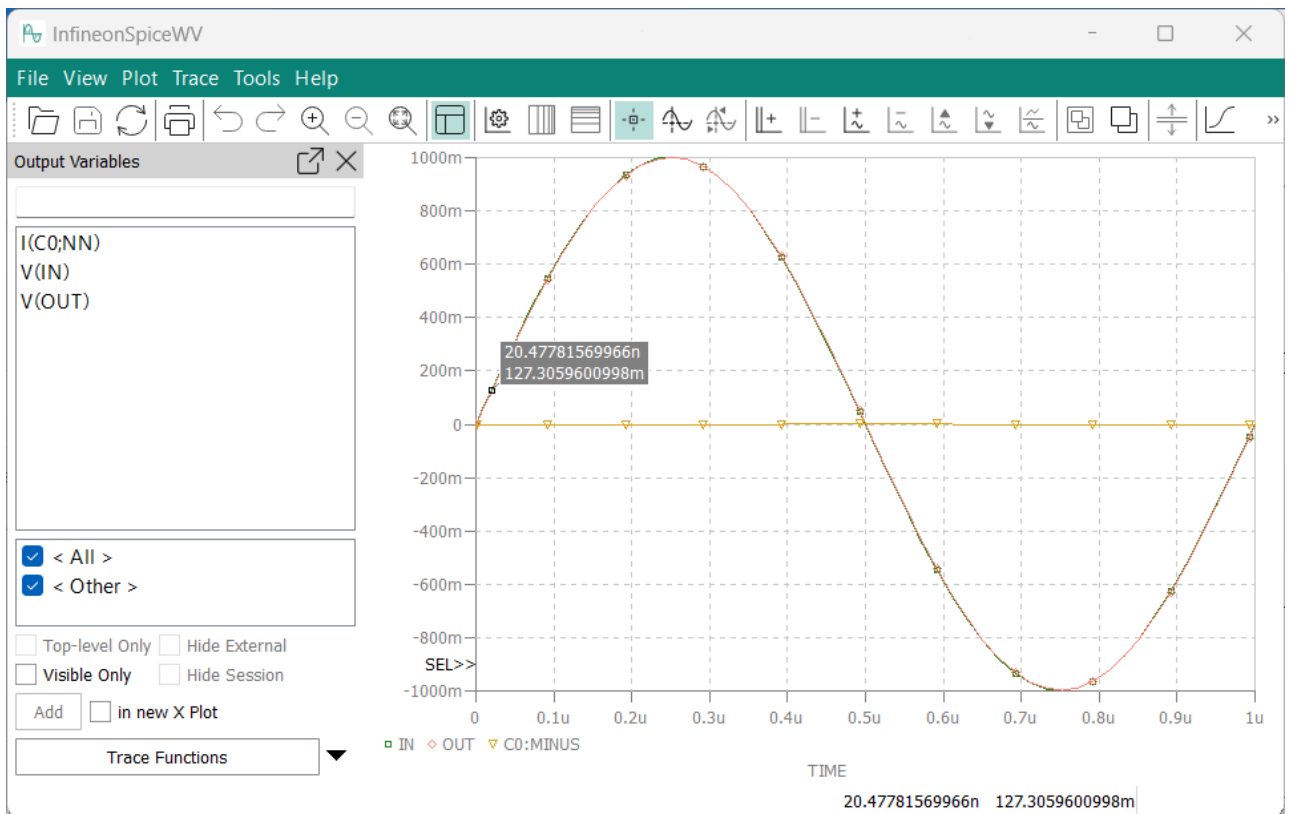
Steps

1. Start the simulation by clicking  in the action bar at the bottom of the **Simulation Environment** view.

InfineonSpice creates a netlist from the schematic and simulates it using the settings in the active simulation profile.

When the simulation is complete, it opens Waveform Viewer. The simulated results are shown as separate traces together in one plot.

2 Getting started



2. On the Waveform Viewer's toolbar, click . This separates the traces into individual plots. Among other things, this gives each plot a separate y-axis (while retaining a common x-axis for all plots) so that you can more easily see traces with comparatively small amplitudes.

If you want to combine the traces again, you can click .

Results

In addition to the traces you can see and work with in the Waveform Viewer, InfineonSpice has generated additional output:

- The **Output** view's **Messages** tab contains a list of status messages that were generated while executing the simulation. These messages are typically of little interest in analyzing the schematic but may be helpful when looking for the sources of unexpected results or outright errors. Understanding them may require technical knowledge of the SPICE simulation engine.
- The netlist, waveforms, and logs that were generated in running the simulation are saved to disk and are available in the project folder in the **Workspace** view.

The name of the folder begins with "sim_", which is followed by the name of the project and the name of the simulation profile that you used. For example, a folder may be named "sim_myProject_myProfile." When you run a simulation again with the same profile, any previous results are overwritten without warning.

If you double-click a log file in the **Workspace** view, it opens in the **Output** view's **Log Summary** tab. You can filter the tab's content by message category.

3 Using InfiniteSpice

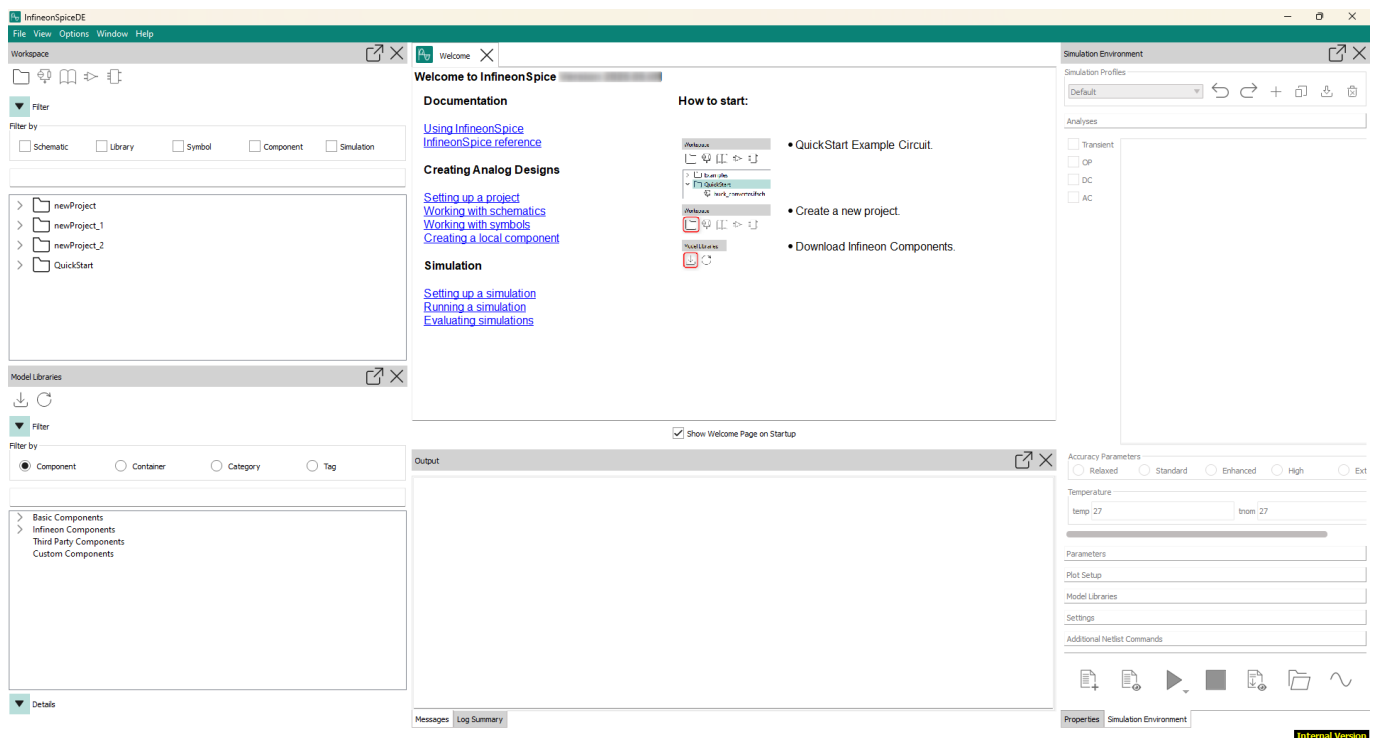
3 Using InfiniteSpice

This chapter provides an overview of the product's features and provides detailed instructions for the usage.

3.1 Work environment

InfiniteSpice has a single window in which tools and data are presented similar to an IDE in tiled specialized views.

The center of the window is the main work area. It contains a tabbed view for each object on which you are working (such as schematics or models).



To the left of the main area are management views:

- The **Workspace** view at the top contains all open projects and their contents. For details, see [Workspace view](#).
- The **Model Libraries** view presents all available standard components that have been made available by Infineon and other sources and that you can use in your circuits. For details, see [Model Libraries view](#).

On the right-hand side are views for displaying and modifying properties:

- The **Properties** view displays detailed information about a selected component and lets you modify it. For details, see [Properties](#).
- The **Simulation Environment** view provides detailed information about the setup used for simulations, which you can also modify. For details, see [Simulation Environment view](#).

At the bottom of the window, the **Output** view provides feedback from the system. For details, see [Output](#).

You can rearrange the views into an configuration that suits your requirements. You can also detach a view from the main window by clicking the Float icon (☐) in the view's title bar. The view becomes a separate window that you can, for example, place on an external monitor. To reattach a detached view, drag it to the window in a location where the other views move aside.

3 Using InfineonSpice

3.1.1 Values and expressions

Values can be entered in as standard, scientific, or quasi-SI notation. They can also be entered as expressions using parameters.

Static values

Static values are usually entered as numbers without SI units (for example, 12m instead of 12mV). Where an SI unit is required, it is derived from the context. For example, if a current source is being configured, the system knows that the value specifies ampere.

Three notations for static values are available:

- Standard numeric notation
For example: 0.03, 2, 1234, 16777216
- Scientific notation
For example: 3e-2, 2e0, 1.234e3, 1.6777216e7
- Spice (quasi-SI) notation
For example: 30m, 2, 1.234k, 16.777216meg

The letters supported in this notation are not case-sensitive:

a	atto	10^{-18}
f	femto	10^{-15}
p	pico	10^{-12}
n	nano	10^{-9}
u	micro	10^{-6}
m	milli	10^{-3}
k	kilo	10^3
meg	mega	10^6
g	giga	10^9
t	tera	10^{12}

SI units are usually not included (for example, 12m instead of 12mV). Where an SI unit is required, it is derived from the context. For example, if a current source is being configured, the system knows that the value specifies ampere.

The decimal point is a period in all notations; thousands separators are not supported.

Expressions

To create values that can be changed by centrally modifying parameters, you can use expressions. Expressions can contain numeric operators (such as + - * / ^), logic operators (such as = <= >) functions, and parameters that have been defined in the **Parameters** tab of the **Simulation Environment** view. For example, if you define a parameter "baseRate", you can specify a value as 2*baseRate+1. If baseRate is set to 5 for a simulation, this expression will result in a value of 11 in the simulation. Details about supported operators and functions can be found in the Parameters and Arithmetic Expressions section of the Titan reference manual.

Expressions are evaluated before each simulation run. Certain advanced kinds of expressions can also be evaluated during a simulation, as described in the Types of Expressions section of the Titan reference manual.

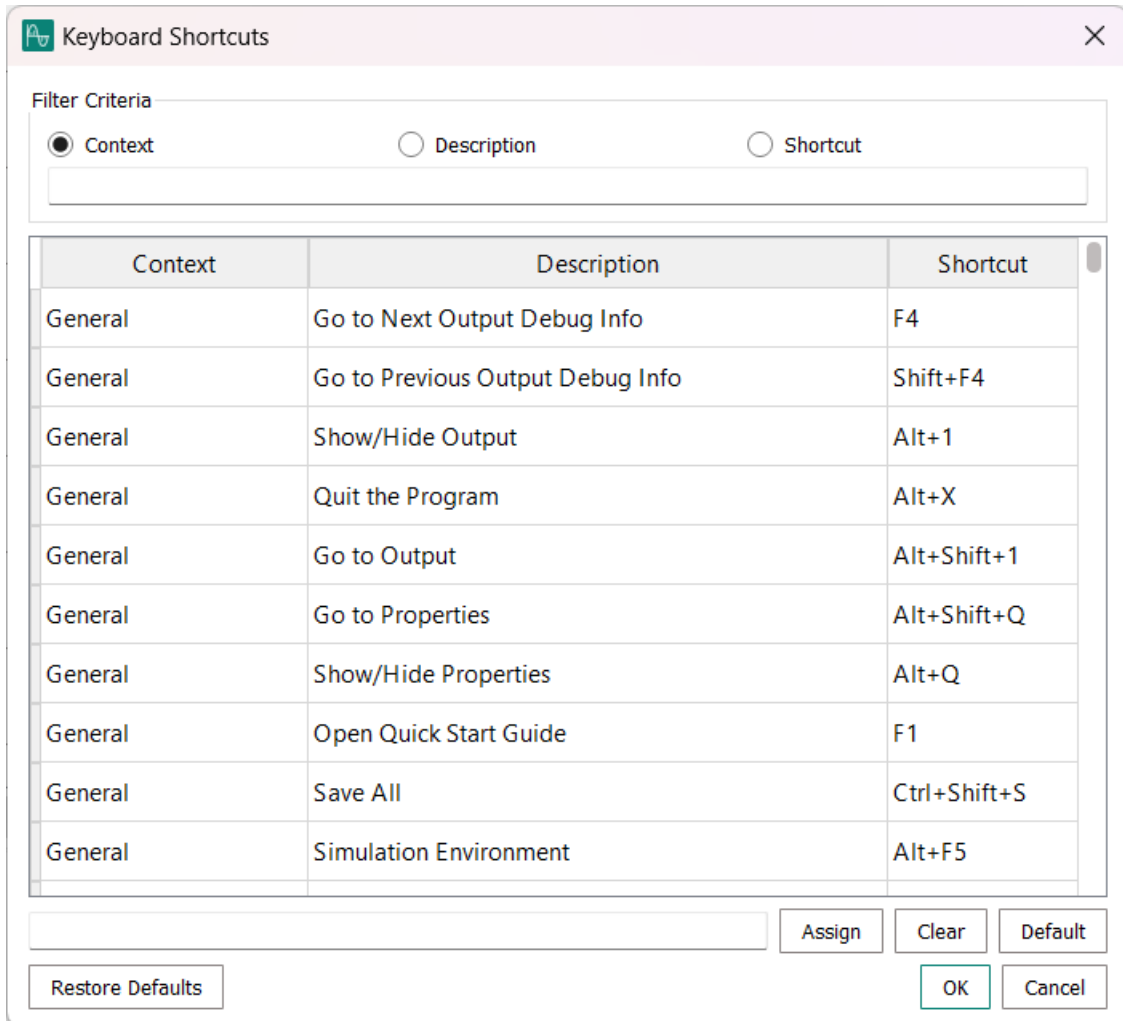
3 Using InfineonSpice

3.1.2 Keyboard shortcuts

The shortcuts that allow control of InfineonSpice and Waveform Viewer from the keyboard can be customized.

Customizing keyboard shortcuts

It is possible to customize keyboard shortcuts for several commands in InfineonSpice and Waveform Viewer. Note that InfineonSpice and Waveform Viewer each have separate sets of shortcuts. This makes it possible to use the same shortcut for different functions in both applications. You can customize shortcuts using the **Keyboard Shortcuts** dialog.



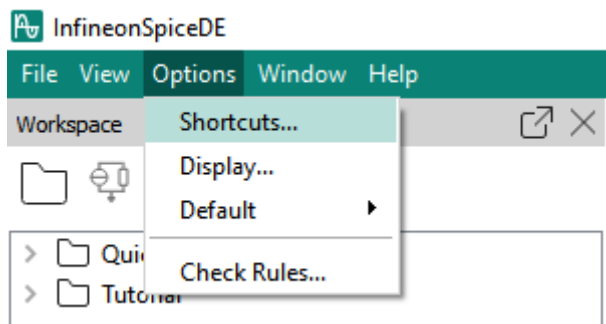
The **Keyboard Shortcuts** dialog box allows users to customize keyboard shortcuts. It features a 'Filter Criteria' section with three radio buttons: **Context** (selected), **Description**, and **Shortcut**. Below this is a search bar. The main area contains a table with three columns: **Context**, **Description**, and **Shortcut**. The table lists various commands and their current shortcuts. At the bottom, there are buttons for **Assign**, **Clear**, **Default**, **Restore Defaults**, **OK**, and **Cancel**.

Context	Description	Shortcut
General	Go to Next Output Debug Info	F4
General	Go to Previous Output Debug Info	Shift+F4
General	Show/Hide Output	Alt+1
General	Quit the Program	Alt+X
General	Go to Output	Alt+Shift+1
General	Go to Properties	Alt+Shift+Q
General	Show/Hide Properties	Alt+Q
General	Open Quick Start Guide	F1
General	Save All	Ctrl+Shift+S
General	Simulation Environment	Alt+F5

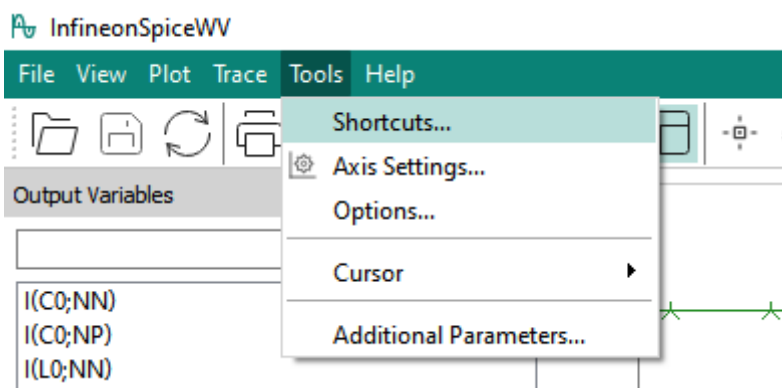
3 Using InfineonSpice

Opening the Keyboard Shortcuts dialog

To open the **Keyboard Shortcuts** dialog in InfineonSpice, choose **Options > Shortcuts**



To open the **Keyboard Shortcuts** dialog in Waveform Viewer, choose **Tools > Shortcuts**



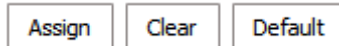
3 Using InfineonSpice

Action overview

The **Keyboard Shortcuts** dialog display a list of available keyboard shortcuts in the application. Some of them are not set by default, but can be customized.

The following controls are located below the list:

- **Assign:** Assign new shortcuts
- **Clear:** Clear new shortcuts
- **Restore Defaults:** Reset shortcuts to their default values



Filtering shortcuts

By default, the dialog displays all available shortcuts, but you can filter them as desired:

1. Choose by which category you want to filter.
In the **Filter Criteria** area at the top of the dialog, click **Context**, **Description**, or **Shortcut**, which correspond to the three columns of the dialog's table.
2. In the text field in the **Filter Criteria** area, enter the text by which you want to filter.

As you type, the display of shortcuts in the table is restricted to those that contain the text you enter in the chosen column. To see all shortcuts again, delete the filter text you have entered.

3 Using InfineonSpice

Assigning a new shortcut

1. Select the desired line of the command from the list
2. Click inside the text box below the list and press the corresponding key combination. The text box will display the shortcut in text form
3. Assign the defined combination of keys to the previously selected command by clicking the **Assign** button

Removing a shortcut

Select the corresponding line in the list and click the **Clear** button.
The command will then not be executable by any shortcut.

Resetting a shortcut

Select the corresponding value from the list and click the **Default** button.

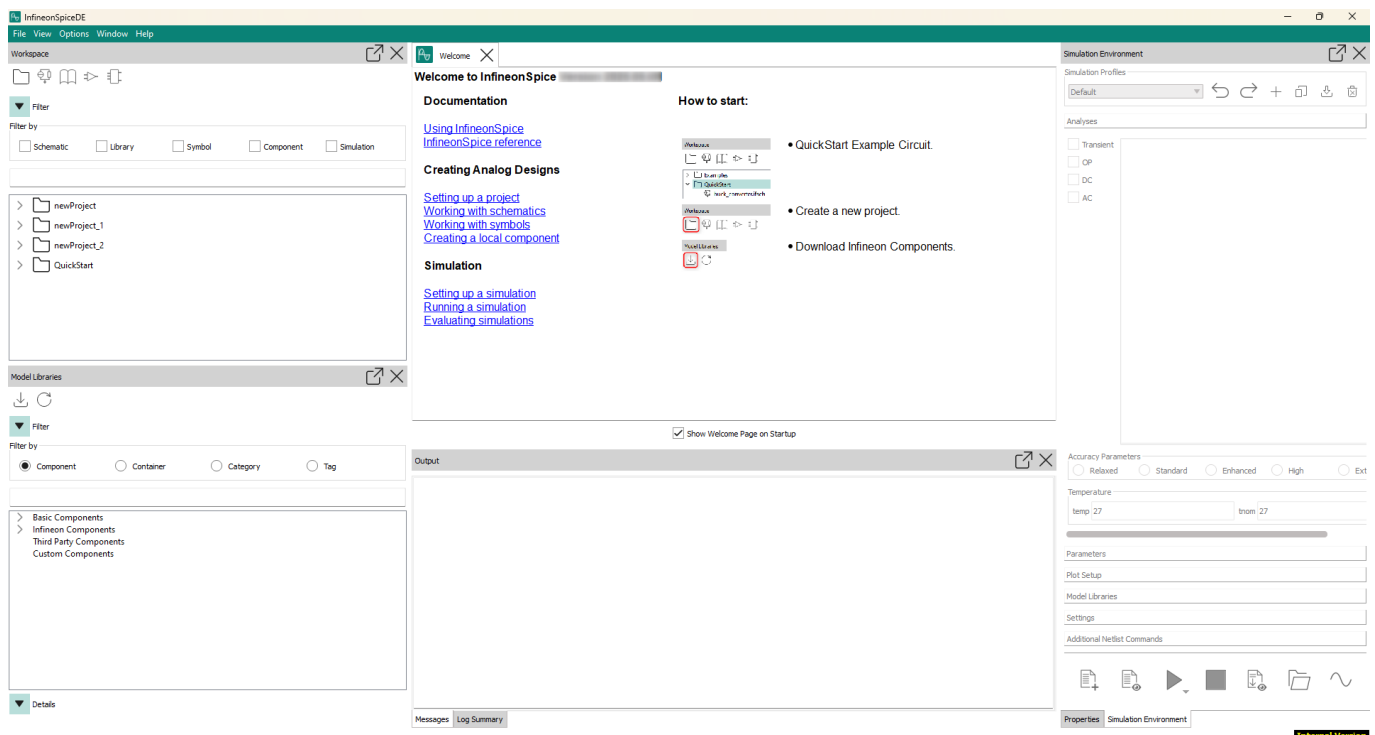
Resetting all shortcuts

Click the **Restore Defaults** button in the lower left corner of the dialog.

3.1.3 View Options

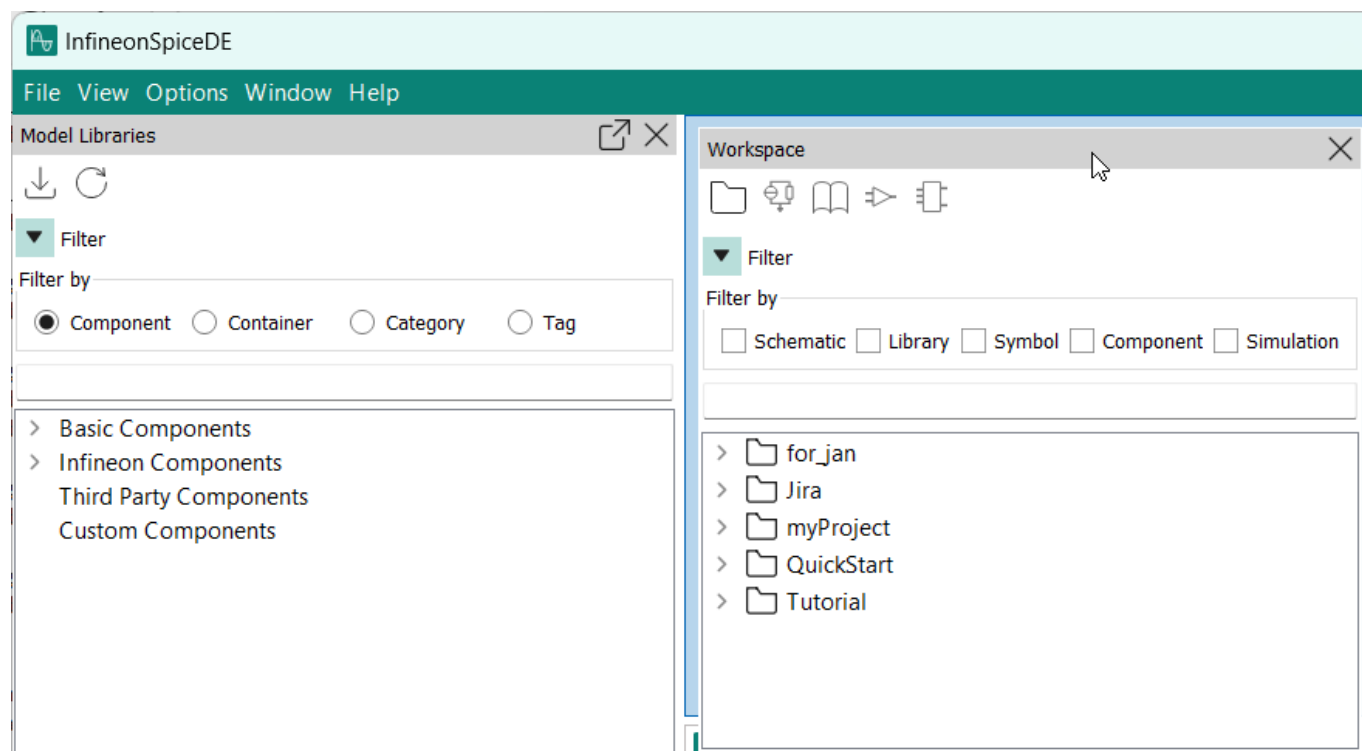
The software offers possibilities to arrange the view layout. It will be described in this chapter.
On initial launch there are several views which can be moved around. You can also show and hide each of it.

- Workspace
- Model Libraries
- Output
- Simulation Environment
- Properties

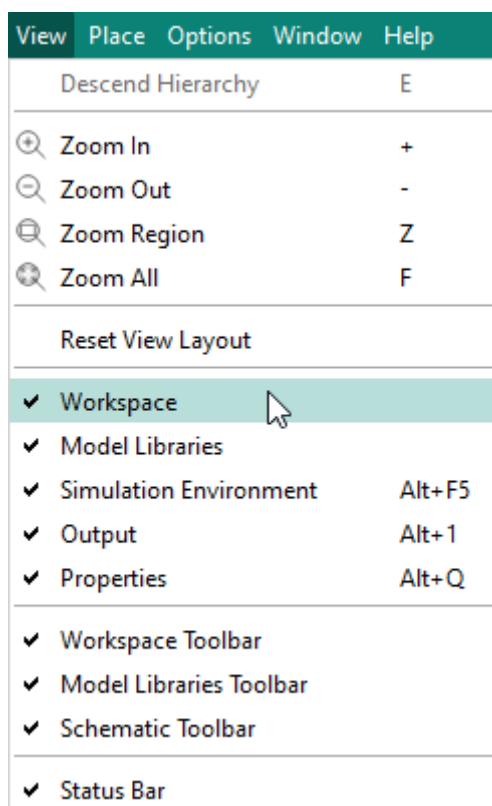


To move a view to another place grab it on the title bar and move it to the favored place. Blue areas indicate where the view will snap in.

3 Using InfineonSpice



You can also close every view and show it again via the **View** menu. Views which are present on the screen are indicated by a tick mark.



The toolbar of the **Workspace** and the **Model Libraries** views can also be shown and hidden via the **View** menu.

In order to reset the layout of the views in the window to its default state, choose **View > Reset View Layout**.

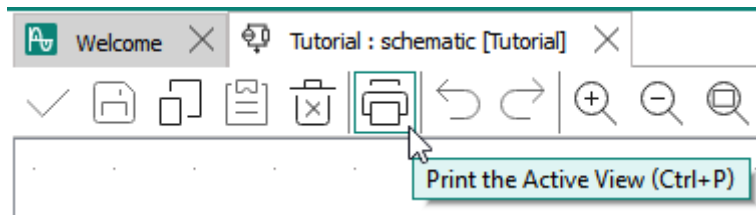
3 Using InfineonSpice

3.1.4 Tips & Tricks

This chapter will provide some hints regarding the usage of the software.

Tooltips

Tooltips occur on almost every button and element in the user interface. Hover with the mouse over it and after a second a Tooltip appears explaining the function of the element. If available the according shortcut will appear in brackets behind the explanation.



Useful Shortcuts

- Check & Save - Checks if the topology of a schematic is valid ('X')
- Fit to Screen - the content of the schematic will be displayed in order to fit to the screen size ('F')

Drag&Drop of parameters in Simulation Environment

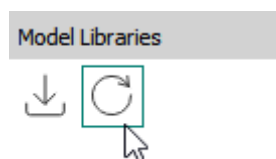
You can change the order of the entries in the **Parameters** table by dragging and placing it to another row. You can also do this for multiple entries if you select more than one.

Mouse middle button moves canvas

If you press and hold the middle mouse button you can move the canvas in the schematic as well as in the symbol editor.

Refresh of 'Model Libraries' view

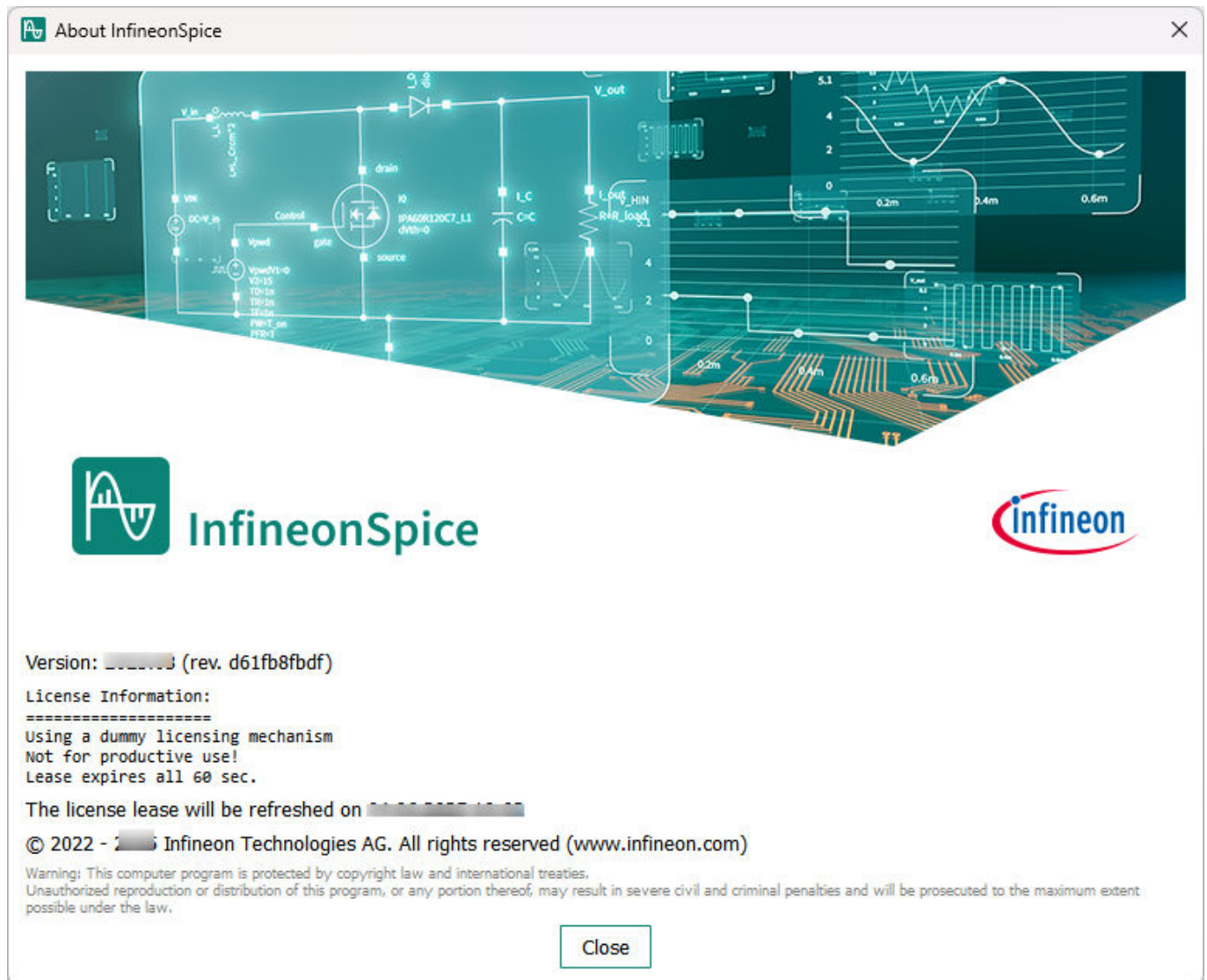
After downloading components or circuits from the Download Manager you can refresh the view with the following button:



Software version

To display information about the software, such as version number or license information go to **Help > About**. In the following screen you can view and also copy the information.

3 Using InfineonSpice



3.2 Project structure

Work in InfineonSpice is organized in projects. A project contains all needed schematics as well as imported or locally created components for use in the circuits.

A project is the fundamental organizational unit when working with InfineonSpice. Projects group everything that you need for a particular undertaking. They can be exported and so exchanged with other users. The **Workspace** view represents a project as a folder .

Each project must contain at least one schematic, which represents a circuit. It can contain multiple schematics, for example, for several development alternatives or if you want to test multiple aspects of a device in simulations.

A schematic contains components that are connected with traces. Components fall into different categories:

- Basic components are generic elements like diodes, resistors, or grounds.
- Infineon components represent electronics products that Infineon offers, such as gate drivers or MOSFETs. InfineonSpice comes with an extensive library of such components that allow you to test them in your designs.
- Custom components can be either ready-made components that sources other than Infineon, in which case they appear in the **Model Libraries** view, or you can create your own local components that are listed in the project's folder in the **Workspace** view.

3 Using InfineonSpice

The behavior of components is described by models, snippets of Spice code, that are contained in libraries. Components are visually represented by symbols; a component is the combination of a model with a symbol.

3.3 Setting up a project

Setting up a project ensures that all required elements for the design are available.

Set up the project when you create a new project or when an existing project is missing some required components.

You can perform some or all of the steps described below when setting up a project.

Steps

1. If the project does not yet exist in the **Workspace** view, create it as described in [Creating a new project](#).
2. If required components are missing, add them to the project.
A new project always contains the basic components that InfineonSpice supports. In addition, you can:
 - Add Infineon components, as described in [Managing Infineon and third-party components](#).
 - Create local components that exist only within the project, as described in [Creating a local component](#).
3. If the project requires more than one circuit, add any missing ones.
Each new project already contains one empty schematic (circuit). You can add schematics in several ways:
 - Create a new schematic as described in [Creating a new schematic](#).
 - Import an existing schematic from a custom container as described in [Custom containers](#).
 - Reuse an existing schematic from another project by copying it as described in [Creating schematics](#).
 - Duplicate an existing schematic in the project as described in [Creating schematics](#).

Results

The project contains the elements you need to work with the circuits.

What to do next

Build the circuits as described in [Working with schematics](#).

3.3.1 Creating a new project

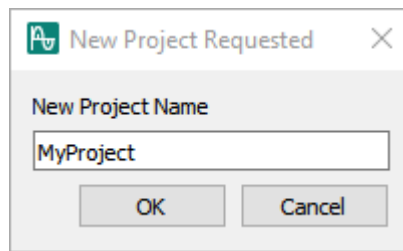
New projects are created by clicking the New Project button.

A new project is typically created to model a new device.

Steps

1. Click the  button at the top of the **Workspace** widget.
You can also press Alt+N or choose **File > New > New Project**.
The **New Project** dialog opens.
2. Enter a name for the new project and click **OK**.
The name may contain letters (a-z), numbers (0-9) and some special characters, such as: # & () + - = @ [] ^ ' { } ~. A dot (.) can also be used except at the end; an underscore (_) can be used except at the beginning of the name. For more information, see Naming directories and files.

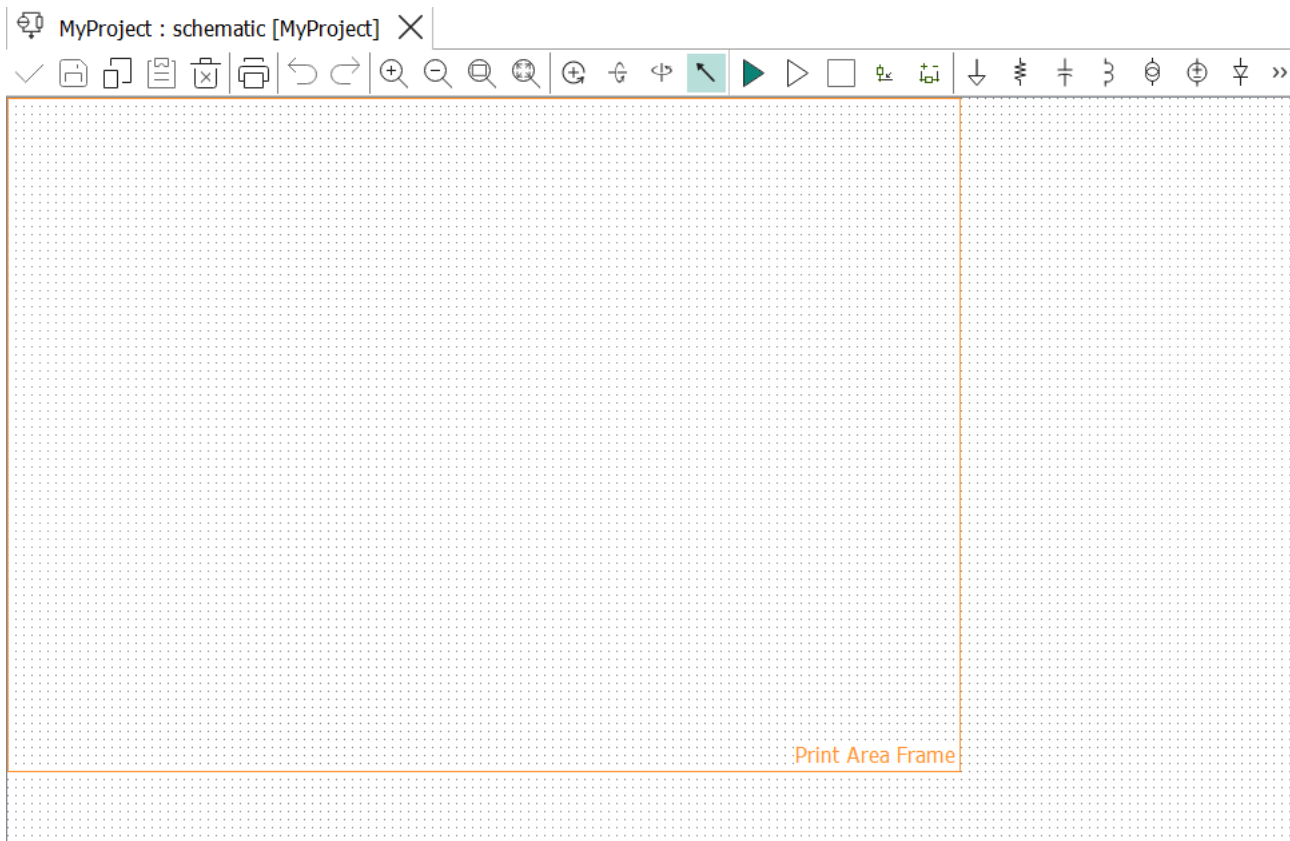
3 Using InfineonSpice



Results

The project appears in your workspace.

The project already contains an empty schematic with the same name as the project.



What to do next

[Creating a new schematic](#)

3.3.2 Setting up new projects

To set a new project, create it in the Workspace view and add the required components.

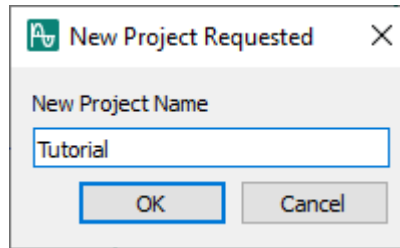
Create a new project to set up the simulation of a new device.

If you want to simulate a device that is similar to an existing device, you can also add set up that device as a separate schematic in the existing project for that device, or you can duplicate the existing project and modify the duplicate as required instead of creating a new project from scratch.

Steps

1. Choose **File > New > New Project**.
The **New Project Requested** dialog opens.

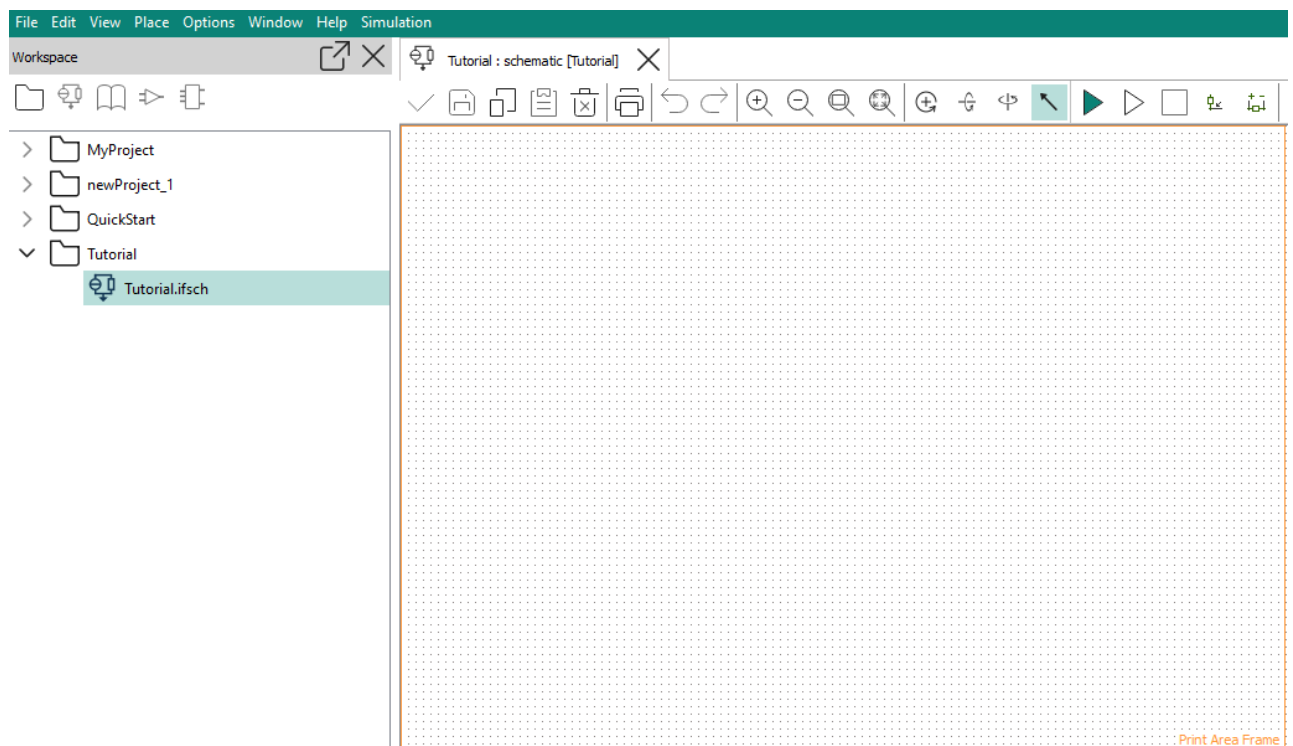
3 Using InfineonSpice



2. Name the project and click **OK**.

A project name can contain letters (a-z and A-Z), digits (0-9), and the special characters # & () + - = @ [] ^ ' {} ~. It can also contain dots (.) but may not end with a dot and underscores (_) but may not begin with an underscore. Refer to Naming directories and files

The new project appears in the **Workspace** view including an empty schematic.



Results

The new project appears in the **Workspace** view. It already contains a schematic, and the tab for that schematic opens in the main work area.

What to do next

Add the components that you need to the project so that you can use them in the schematic.

3.3.3 Managing project contents

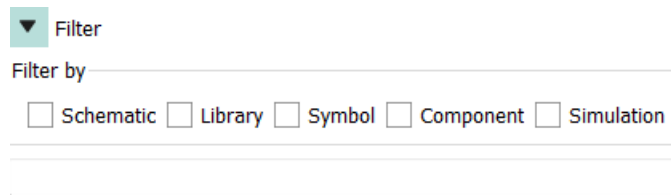
Elements of a project – such as schematics, logs, or waveforms – can be deleted, renamed, copied, and managed in other ways.

Filtering workspace content

In large projects, it can be difficult to find the element you want to work with. This can be made easier by filtering the contents of the **Workspace** view.

To filter the content, click the Filter triangle below the workspace toolbar. This expands the filtering section of the **Workspace** view:

3 Using InfineonSpice



Filter

Filter by

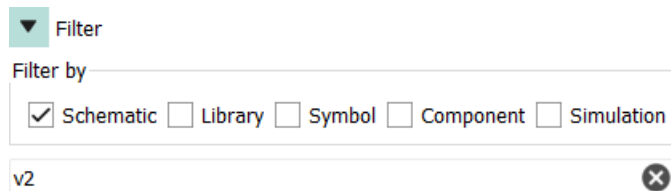
☐ Schematic ☐ Library ☐ Symbol ☐ Component ☐ Simulation

If you want to display only certain types of elements (such as schematics or symbols), check those element types. If you do not check any element types, all types of elements are displayed. (Note that project folders are always displayed, regardless of the checked options.)

If you want to display only elements with certain names, enter text into the text field below the checkboxes. Only elements are displayed whose names contain the entered text.

If you both check element types and enter text, only elements of the selected types with the specified text in their names are shown.

For example, these filter settings display only schematics whose names contains "v2":



Filter

Filter by

☒ Schematic ☐ Library ☐ Symbol ☐ Component ☐ Simulation

v2

Creating new elements

New elements are created via the buttons in the Workspace view toolbar. You can create schematics, libraries, symbols, and local components in this way.

Result folders of a project and the elements in them are created by running a simulation; they cannot be created manually. This applies to waveforms, netlists, and log files.

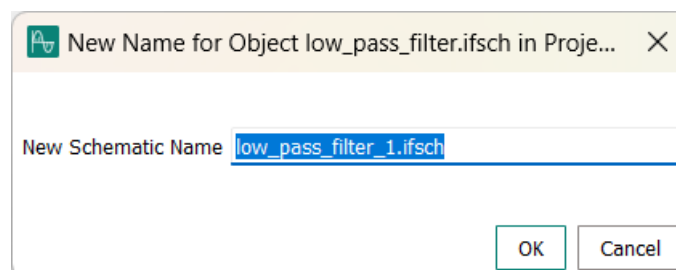
Deleting elements

Elements are deleted by right-clicking them and choosing **Remove** from the context menu.

It is not possible to delete individual elements of a result folder, but the result folder can be deleted as a whole.

Renaming elements

To rename an element, right-click it and choose **Rename** from the context menu. This opens the **New Name for Object** dialog:



New Name for Object low_pass_filter.ifsch in Proje... X

New Schematic Name low_pass_filter_1.ifsch

OK Cancel

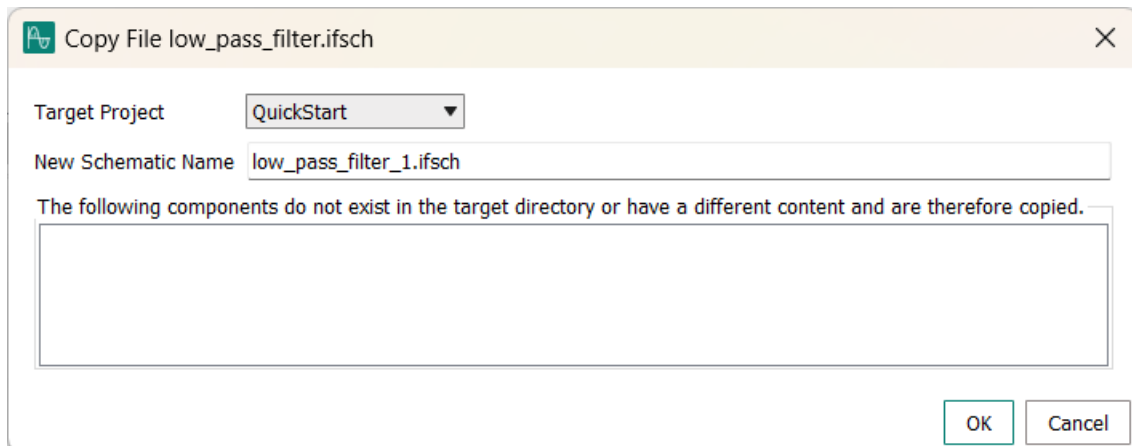
Enter the desired name and click **OK**. Make sure not to change the filename extension when you rename an object.

You can rename schematics, libraries, symbols, and local components in this way. Result folders and their contents cannot be renamed.

Copying elements

To copy an element to another project, right-click the object and choose **Copy** from the context menu. This opens the **Copy File** dialog:

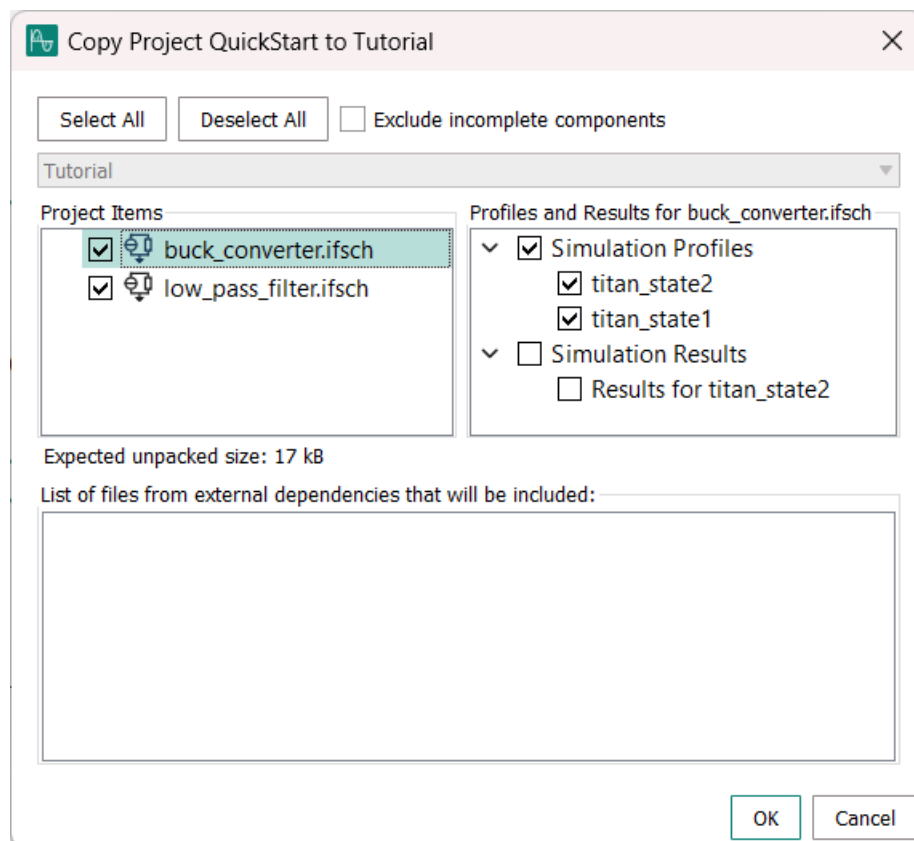
3 Using InfineonSpice



Choose the project to which you want to copy the element from the **Target Project** list and click **OK**. If you choose the same project from which you copying, the element is duplicated in its project.

Instead of right-clicking, you can also drag the element to the desired project. This opens the **Copy File** dialog with the target project already selected.

To copy multiple elements in one step, you can drag one project folder on top of another. This opens the **Copy Project** dialog:



Check the project elements that you want to copy to the target project, uncheck the elements that you do not want to copy, and click **OK**. All checked elements are copied.

Duplicating an element

A element can be duplicated by copying it to the project in which it is contained. See the description of copying element, above, for details.

3 Using InfineonSpice

Accessing the file of an element

If you want to access the file of an element, for example, for copying it to a different system or opening it with an external tool, you can do so by right-clicking it and choosing **Show in File Explorer** from the context menu. That opens the folder on disk that contains the file.

3.3.4 Managing Infineon and third-party components

Which Infineon and third-party components are available to your projects is managed in the Download Manager.

The online Model Store contains representations for a wide range of Infineon and third-party components which you can import into InfineonSpice to make them available for all of your projects.

Use the Download Manager to manage the Infineon and third-party components on your workstation by adding components of interest, removing those that you no longer want to use in your designs, and updating already downloaded component models when new revisions become available.

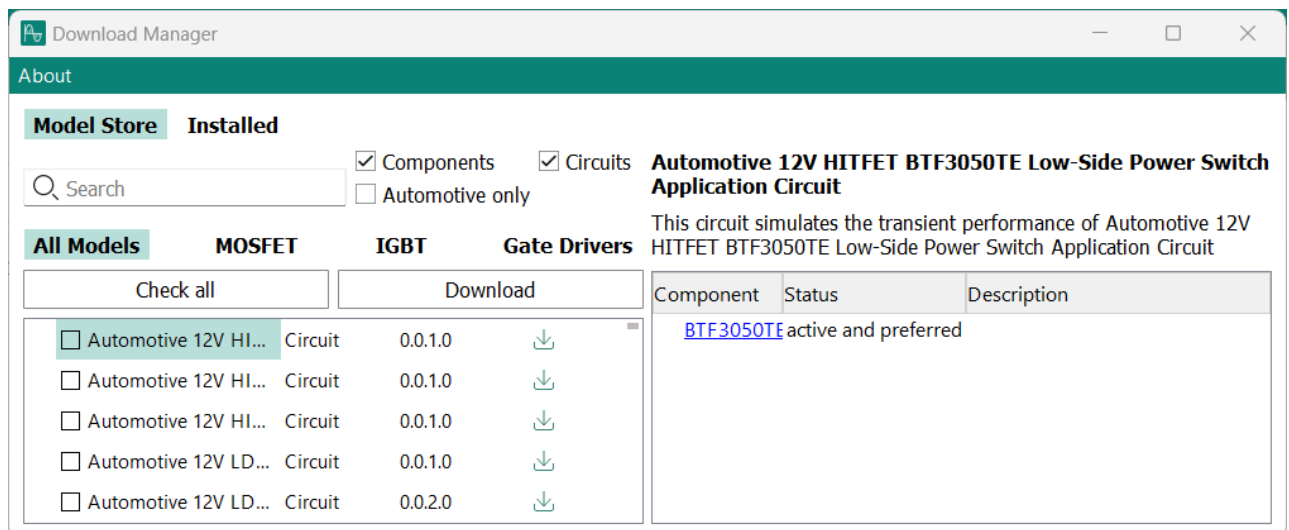
Note that the Download Manager provides access to a limited range of third-party components that complement Infineon components and have been vetted by Infineon. You can manually create additional components by importing the respective model libraries as described in [Working with libraries](#) and creating the desired components from them as described in [Creating a local component](#).

Steps

1. Open the Download Manager by clicking the  button in the **Model Libraries** view.

You can also press Ctrl+Alt+D.

The **Download Manager** window opens.



For an overview of the Download Manager, see [Working with the Download Manager](#).

2. If you want to import additional components, select and download them.
 - a. Select the containers that contain the desired components.

To select an individual container, check it in the list of containers on the left. Click a container to see the components it contains in the component list on the right.

To select all components in the list, click the **Check All** button.

You can filter the displayed components in various ways:

 - To view only a particular product category, click the appropriate tab (such as **MOSFET**).

3 Using InfineonSpice

- To view components, check **Components**; to view circuits, check **Circuits**. To hide components that do not have automotive applications, check **Automotive only**.
 - To filter the display of containers in the list, enter text into the "Search" field at the top. The component list displays only containers whose names include the filter text.
- b.** Click Download.
- This downloads all selected component models to your local copy of InfineonSpice. They are listed in the Infineon Components section of the Model Libraries and are available in all projects.
- 3.** If you want to delete a downloaded container, click its Delete button ().
- a.** Click the **Installed** tab.
- While you can delete downloaded components in the **Model Store** tab in the same way, doing so in the **Installed** tab is usually easier.
- b.** If desired, filter the displayed list of containers as described above.
- c.** For each container that you want to remove from your local installation, click the  button to the right of its name. (If the  button is shown, click that first and then .)
- This removes the container and all its components from your copy of InfineonSpice. If you want to reverse this decision later, you can do so by downloading the container again.
- 4.** If you want to update component models you have already downloaded, click their Update buttons ().
- a.** Click the **Installed** tab.
- While you can update downloaded components in the **Model Store** tab in the same way, doing so in the **Installed** tab is usually easier.
- b.** If desired, filter the displayed list of containers as described above.
- c.** For each container that you want to update, click the  button to the right of its name.
- This downloads the most recent component models for all components in the container. All placed versions of this component in projects are updated, as well.
- If no  button is shown for a container, no update for it is available in the model store.

Results

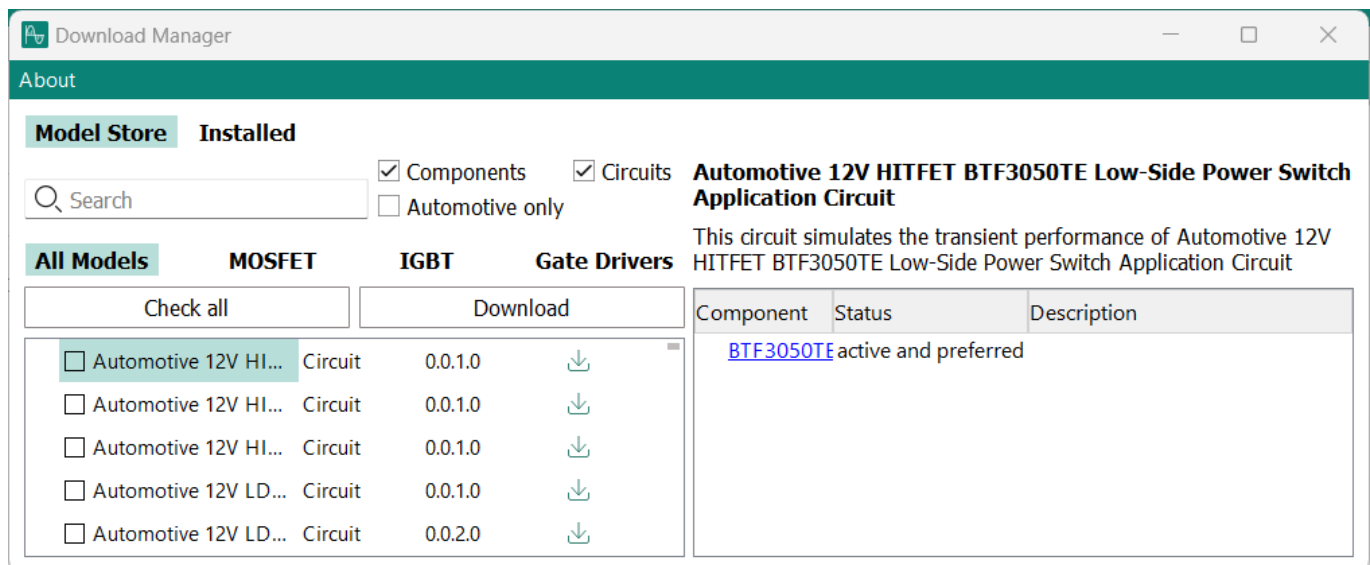
You have adapted the selection of Infineon component models available in your copy of InfineonSpice.

3.3.4.1 Working with the Download Manager

Using the Download Manager, you can manage the Infineon components that are installed on your system and download additional ones.

You access the Download Manager through the **Download Manager** window. To open the window, click the  button in the **Model Libraries** view.

3 Using InfineonSpice



The window contains two main panels, which are both similar: The **Model Store** panel, where you can download packages, and the **Installed** panel, where you can update and delete packages that are already present on your system. To switch between the panels, click the respective tab at the top left.

Below the two tabs are the controls for searching and filtering the displayed packages.

The main part of the window is the list of packages. Each entry in the list represents a package. For each package, the name, type, and version are displayed, and a button is provided using which you can (depending on the panel) download, update, or uninstall the package.

To the right of the list is a sidebar that displays the description of the selected package and the components that are included in it.

Searching for packages

To search for a component, enter the text that you want to search for (such as the product number of a component) in the search field at the top left of the window. The list of displayed packages is dynamically reduced to only those packages that contain the text in their name, their description, or the name of a contained component.

Note that all filters (see below) remain active during a search. For example, if you have checked **Automotive only**, searching finds only automotive packages.

Filtering packages

If you are looking for packages of a certain type or application area, you can filter the displayed list of packages:

- To filter the packages by type, check either **Components** or **Circuits**.
At least one of these options must be checked; you cannot uncheck both.
- To display only packages for automotive applications, check **Automotive only**.
- To display only packages from a certain category, click the **MOSFET**, **IGBT**, or **Gate Drivers** tabs.
To see packages of all categories, click the **All Models** tab.

The number at the end of the tab indicates the number of packages in the category that match the current filtering criteria (components, circuits, automotive) and search string, if any.

Installing packages

Installing packages is done in the **Model Store** tab.

To install a single package, click the Install button () in the package's list entry. If there is an Uninstall button () or Update button () instead of the Install button, the package is already installed on your system.

3 Using InfineonSpice

To install multiple packages, indicate all desired packages by checking them in the list:

- | | | | |
|-------------------------------------|---|--------------------|---|
| ☒ | MOSFET 100V N-Channel IR MOSFET - IRFB... | Components 1.0.0.0 |  |
| ☒ | MOSFET 100V N-Channel IR MOSFET - IRF... | Components 1.0.0.0 |  |
| ☐ | MOSFET 100V N-Channel IR MOSFET - IRFI... | Components 1.0.0.0 |  |
| ☒ | MOSFET 100V N-Channel IR MOSFET - IRFL... | Components 1.0.0.0 |  |

You can also click the **Check All** button above the list to indicate every package that is displayed in the list. After you have indicated the desired packages, click the **Download** button above the list.

Updating packages

Installed packages can be updated in both tabs of the window.

When an update is available for an installed package, the Update button () is displayed beside the package in the list. To update the package, click the button.

If the Update button is not displayed, the most recent version is already installed on your system.

Uninstalling packages

Installed packages can be uninstalled in both tabs of the window.

To uninstall a package, click the Uninstall button () beside the package in the list.

If an Update button () is displayed instead of the Uninstall button, check the checkbox at the left of the package entry, make sure that no other packages are checked, and then click the **Uninstall** button above the list.

To uninstall multiple packages, check all packages that you want to uninstall. You can also click the **Check All** button above the list to check every package that is displayed in the list. After you have indicated the desired packages, click the **Uninstall** button above the list.

3.4 Working with schematics

Schematics can be configured much like real-world circuits. When they are finished, their operation can be simulated.

Creating schematics

Each project comes preconfigured with an empty schematic. To create additional or replacement schematics, select the project in which the schematic is needed in the **Workspace** view and click the  button in the toolbar. For details, see [Creating a new schematic](#).

Instead of creating an empty schematic, you can also create a duplicate of an existing schematic by right-clicking it and choosing **Copy** from the context menu.

In the same way, you can copy a schematic from another project. To do so, right-click it in the project in which it already exists, choose **Copy** from the context menu, and specify the new project as the target.

Populating a schematic

To edit a schematic, right-click it in the **Workspace** view and choose **Open in Schematic Editor** from the context menu. This displays the schematic in a separate tab in the center of the main program window.

You can add all types of components to a schematic by right-clicking them in the **Model Libraries** view and choosing **Place** or by double-clicking them. This causes an outline of the component to appear under mouse when you move the mouse over the schematic tab. Click to place a copy of the selected component; press Esc to stop placing this type of component. Clicking the right mouse button before placing a component rotates it 90° counterclockwise without placing it.

Basic components are also available in the toolbar, along with wires and solder dots:

3 Using InfineonSpice



The Resistor () , Current Source () , and Voltage Source () tools let you choose between several different types of the respective component.

Note: We recommend that you place the terminals of the components on the grid (the gray dots in the background of a schematic tab) to help with proper wiring.

When you have inserted a component, right-click it and choose **Properties** from the context menu. This opens the **Properties** dialog where you can specify the name and parameters for the component. Mandatory fields are highlighted in yellow. Definitions of the parameters are available by hovering the mouse pointer over the parameter's label. You can enter numbers in standard notation (for example, 1000), scientific notation (1e3) or using SI-like prefix letters (1k). You can also use expressions, with or without parameters. For details, see [Values and expressions](#).

To connect components, use wires ( or ) between the components' connectors. Do not use lines () for this purpose, they are ignored by the simulation. When wires cross, solder dots are placed automatically. To add solder dots on connections where they have not been created automatically, use the Solder Dot tool () . To remove a solder dot, drag one of the wires away from the crossing using the Select Mode () tool.

You can also use the Select Mode to reposition components and wires.

To delete components or wires, select them (using the Select Mode tool ) and press the Del key or right-click them and choose **Delete** from the context menu.

If you want to leave a connection intentionally open, we recommend that you attach the open end to the NC component which is found in the Basic Components section of the **Model Library** view.

Annotating a schematic

To annotate a schematic, you can:

- Add a name to a wire using the Wire Name tool () . Note that two wires with the same name are connected internally in the netlist, even if they have no connection in the graphic of the schematic.
- Change the label of a placed component by entering the desired label in the **Name** field in its **Properties** dialog.
- Add any desired text in any location using the Text tool () .
You can change the font, size, color, alignment, and rotation of placed text in the **Text** dialog, which you can open by selecting the, right-clicking it, and choosing **Properties** from the context menu.
- Add lines and rectangles using the Line () and Rectangle () tools.
You can change the line styles, colors, and (for rectangles) fill styles in the **Properties** dialog. You can rotate lines and rectangles using the Rotate tool () .

Except potentially for the names of the wires, these annotations have no effect on the electrical properties of the schematic or the results of the simulation.

Editing read-only schematics

The Infineon libraries that are shown in the **Model Libraries** view contain (in addition to components) schematics. These schematics are read-only, that is, you cannot edit the versions in the library. You may also encounter read-only libraries in other cases.

While you cannot edit these schematics in place, you can still create copies of them that you can modify. To do so, copy the schematic you want to modify to a new or existing project by right-clicking it and choosing **Copy to New Project** or **Copy to Existing Project**, respectively, from the context menu. In the **Workspace** view, open the copy of the schematic in the project folder and edit it as you would any other schematic.

3.4.1 Creating a new schematic

New schematics are created by clicking the New Schematic button.

Schematics represent circuits in InfineonSpice.

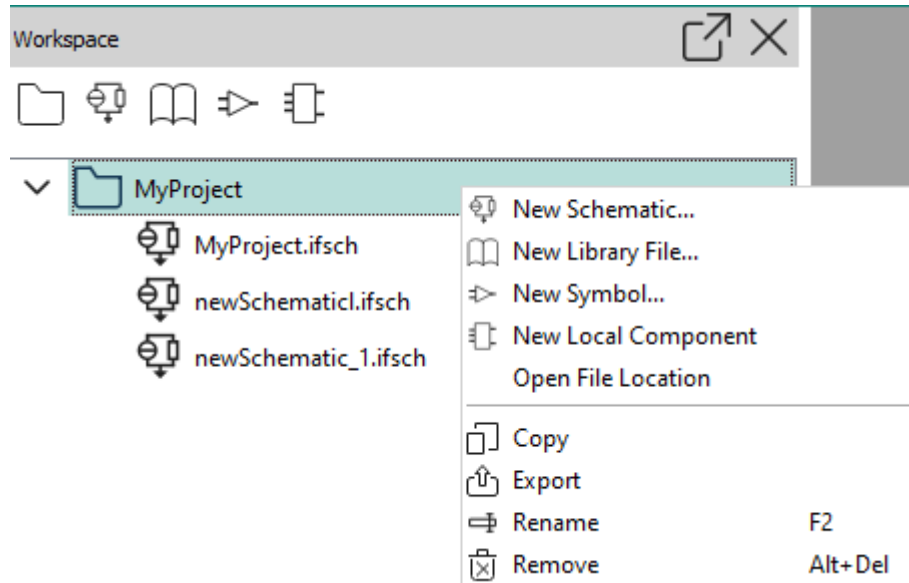
3 Using InfineonSpice

Each project already contains an empty schematic when created. Create a new schematic in a project when the project requires an additional circuit.

Steps

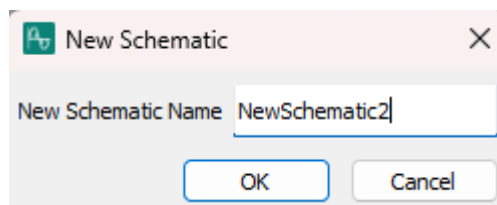
1. Select the relevant project in which you want to create the schematic and click the  button in the **Workspace** widget.

You can also right-click the project and choose **New Schematic** from the context menu or select the project and choose **File > New > New Schematic**.



The **New Schematic** dialog opens.

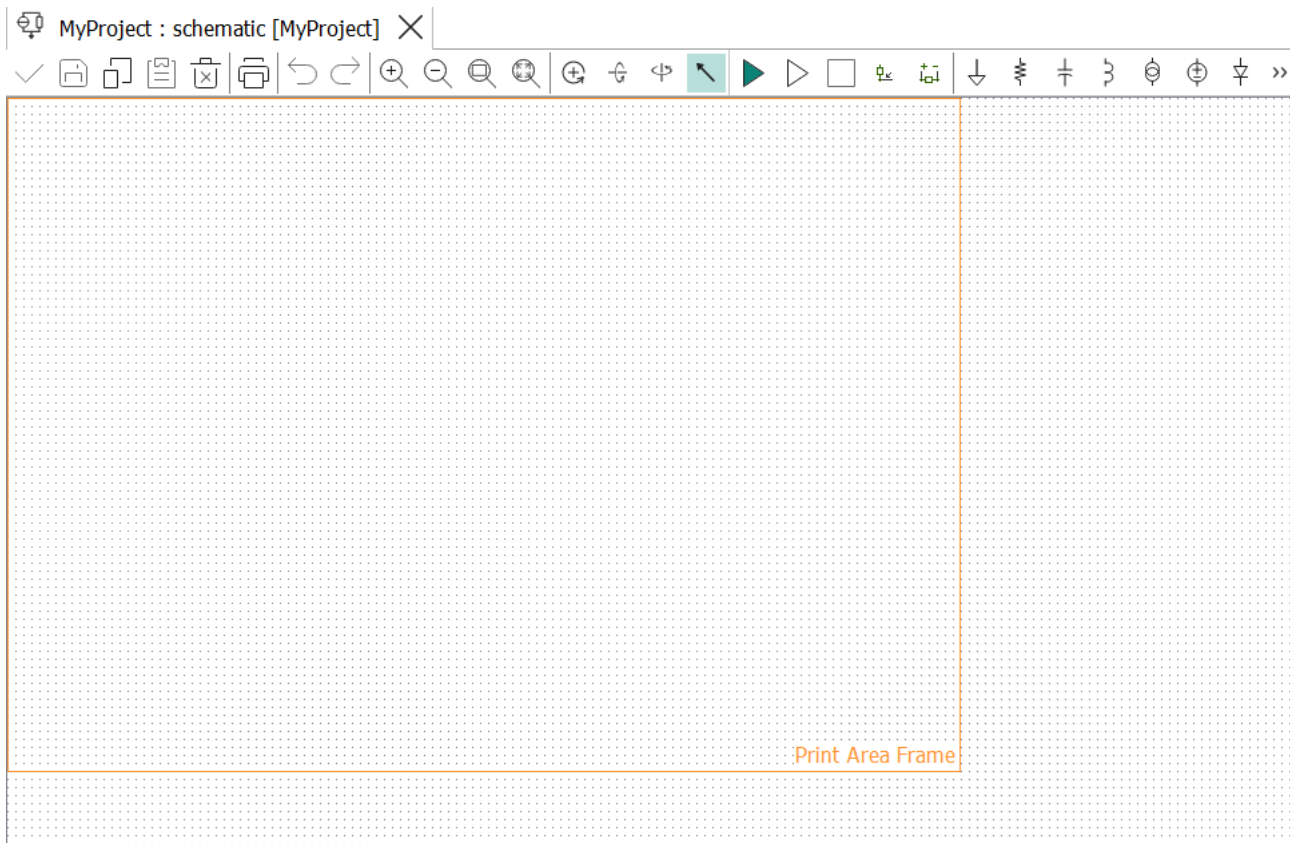
2. Enter a name for the new schematic and click **OK**.



Results

The schematic appears in your workspace.

3 Using InfineonSpice

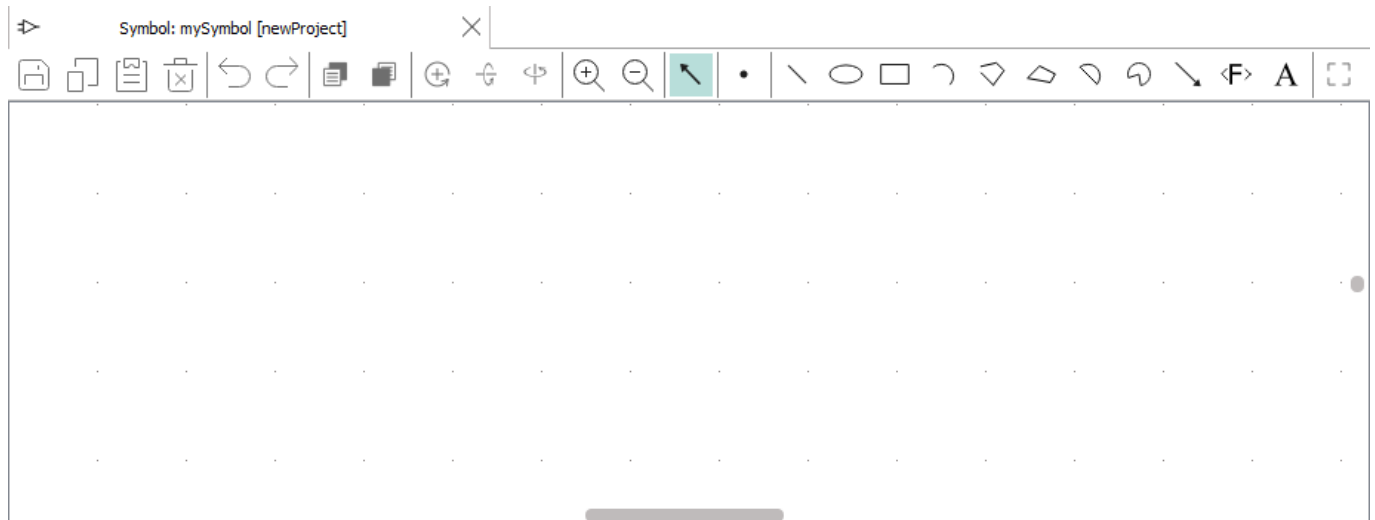


3.5 Working with symbols

Symbols represent components in schematics. You can create and modify symbols for local components in the symbol editor.

Creating symbols

To create a new symbol, you can right-click the project to which you want to add it and choose **New Symbol** from the context menu. (See [Creating a new symbol](#) for more information.) After you have named the symbol, the symbol editor opens with an empty canvas.



To create the symbol, draw the desired graphical representation with the graphics tools in the toolbar:



3 Using InfineonSpice

The tools are the standard tools that you can also find in other drawing programs and work in a similar way. Only the Substitution tool (<F>) is unusual: With it, you can insert placeholder text fields in the symbol drawing that get replaced in the diagram by the respective values for the component.

Pins

To make the symbol usable in a local component, you also must add the pins of the component. They are the locations where connections to other components can be made in the schematic. A symbol must have exactly as many pins as specified by the model with which it will be paired to create a local component. Pins are added with the SetPin tool (•).

When you add a pin using the SetPin tool, you first specify the pin's properties in the Attributes dialog. To avoid future difficulties, you should specify the same name as the pin has in the model, and you should correctly set the direction. If you do not want to show the pin names in the schematic, uncheck **Visible**.

Bounding box

Add the bounding box (the area where a user can click in the schematic to select the device represented by the symbol) by clicking  in the toolbar and choosing **Automatic** in the displayed dialog. (While it is possible to manually specify a different bounding box, we recommend that you always use the Automatic option.)

Editing symbols

You can edit an existing symbol by double-clicking its .ifsym file in the **Workspace** view. That opens the symbol in the symbol editor, where you can modify it as described above.

To edit the symbol of an existing local component, right-click the local component in the **Workspace** view and choose **Open in Symbol Editor** from the context menu, which also opens the symbol in the symbol editor.

3.5.1 Creating a new symbol

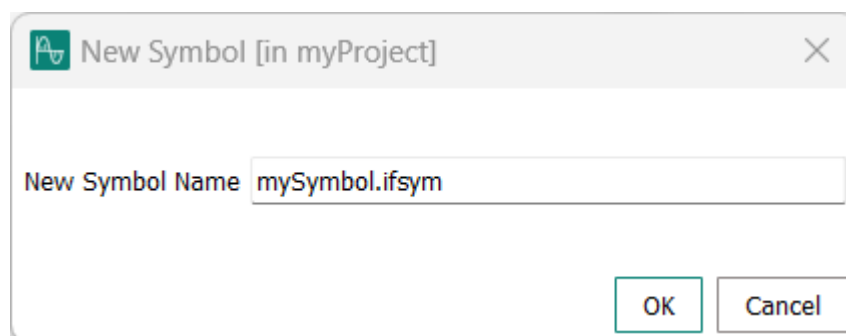
New symbols are created by clicking the New Symbol button.

Symbols are graphical representations of components in the schematic editor. Create a new symbol when you want to create a new local component for use in schematics or when you need a different representation for an existing local component.

If you want to create a new local component, note that a matching model in a library contained in the same project is required in addition to the symbol. If you create a local component, you can automatically create a symbol for a model in your project. (See [Creating a local component](#).)

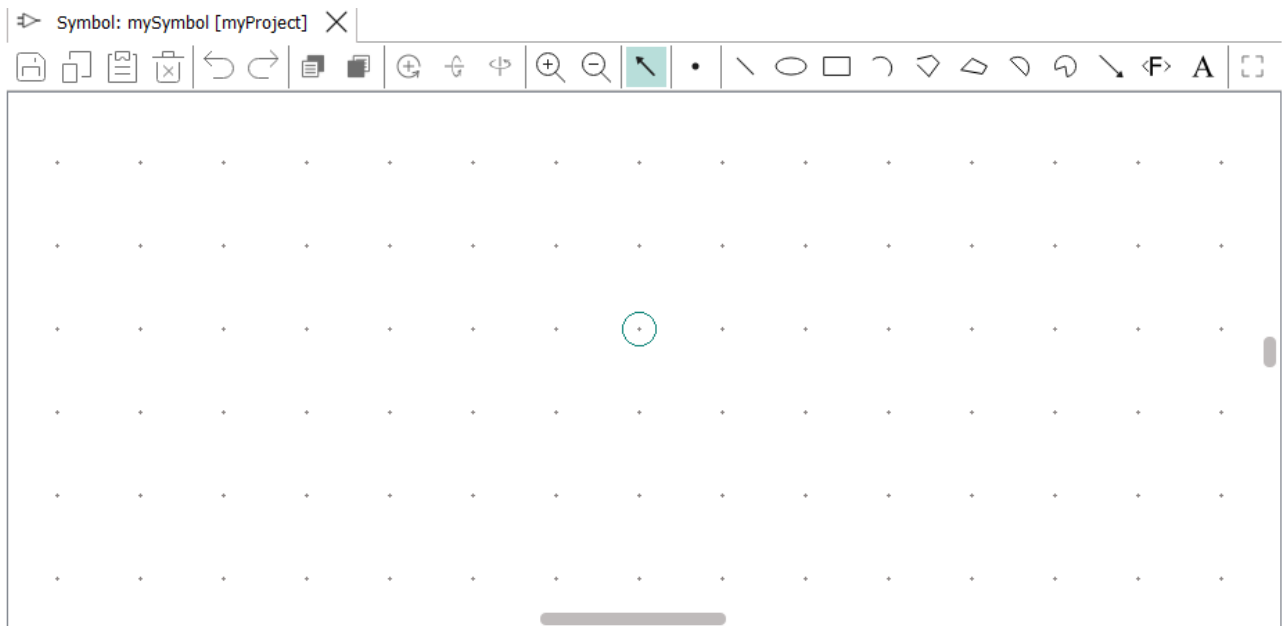
Steps

1. Select the project to which you want to add a symbol and click the  button in the **Workspace** widget. You can also right-click the project and choose **New Symbol** from the context menu. The **New Symbol** dialog opens.
2. Enter a name for the new symbol and click **OK**.



The symbol file appears in your workspace and the symbol editor opens with the new file ready for editing.

3 Using InfineonSpice



The circle in the drawing area indicates the location that will be placed where you click the mouse when using the symbol. Usually, it is best to draw the symbol so that this circle is at the center of the symbol.

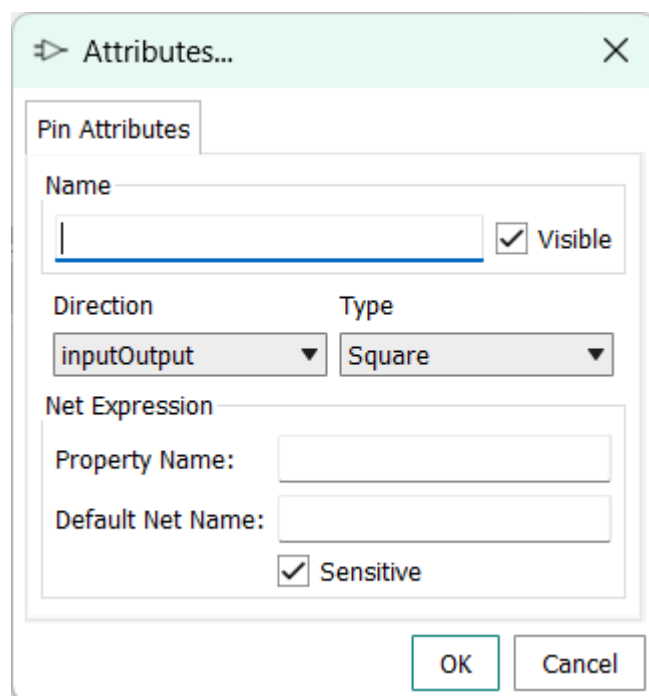
3. Place the pins for the component that the symbol will represent.

- a.** Choose **Options > Display**, click the **Grid** tab, and click the **Default** button.

Doing so resets the grid to its default settings. We strongly recommend to always place pins using the default grid settings and making sure that the **Snap to Grid** option in the **Grid** tab is active. If you use other grid settings, it may be difficult to correctly connect wires to the devices, and hard-to-detect errors may occur.

You can skip this step if you are sure that the symbol editor is already set to the default grid settings.

- b.** Choose the Set Pin tool (•).
The **Attributes** dialog opens.



- c.** Specify the pin attributes and click **OK**.

3 Using InfineonSpice

You must always specify the pin name in the **Name** field. This name must match the name of the pin in the model that describes the component's behavior. If you do not want the name to be visible in the symbol, uncheck the Visible option.

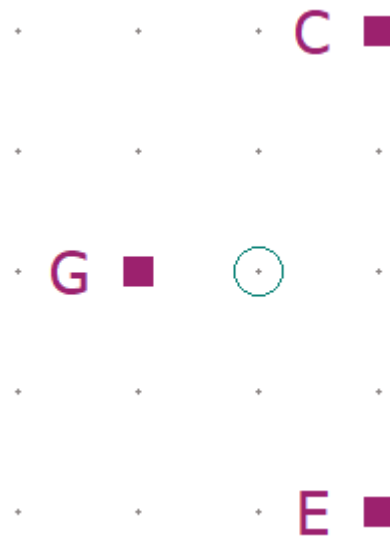
You also must choose the correct direction from the **Direction** list.

The other settings in the dialog are optional.

The mouse pointer changes shape according to the name and shape you have specified.

- d. Click to place the pin.
- e. Repeat the preceding the preceding steps until you have placed all pins.

For example:

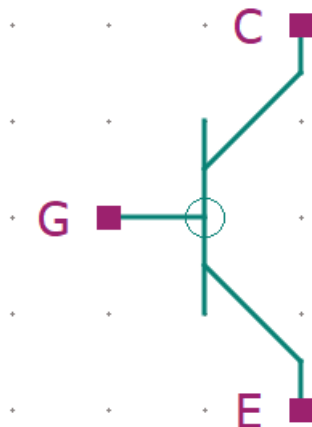


4. Draw the desired symbol using the graphics tools at the right of the toolbar.

The tools work similar to the tools available in common graphics applications. Note that each tool remains selected until you select another tool or press Esc.

You can reposition the pins during this process. If you do so, make sure that the default grid is still active and that **Snap to Grid** is switched on to avoid misalignment issues when using the local component.

For example:



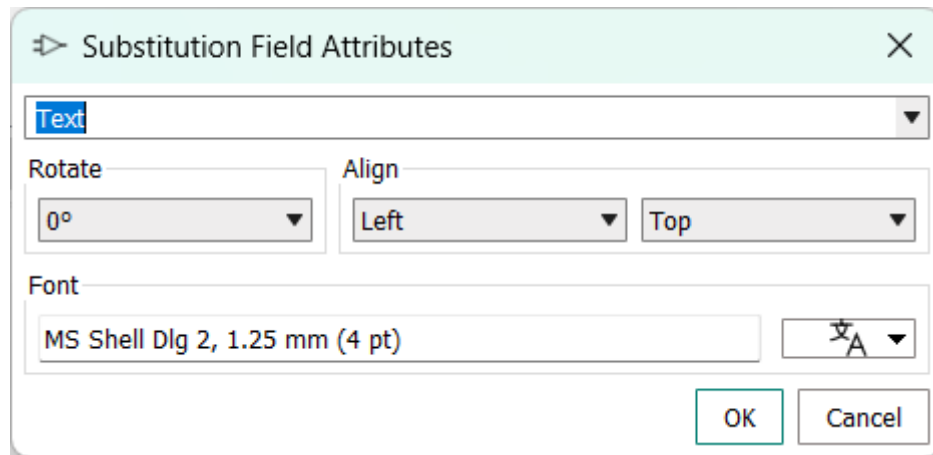
3 Using InfineonSpice

5. If desired, place text or placeholders for component properties in the symbol using the Substitution Field (**<F>**) tool.

Depending on the type of label you specify, the text is either displayed verbatim or replaced with the actual values of a property when the local component is placed in a schematic.

- a. Click the **<F>** tool.

The **Substitution Field Attributes** dialog opens.



6. Save the changes to the symbol.

What to do next

Use the symbol to create a local component as described in [Creating a local component](#).

3.6 Creating a local component

Local components are custom parts that you can use in schematics. They are a combination of a netlist subcircuit and a symbol.

To create local component, its subcircuit must be described in the netlist of a library that is contained in the project. A symbol is also required, which can be created before or during the creation of the local component.

The content and structure of InfineonSpice libraries follows standard SPICE conventions, as described in [Working with libraries](#). The description of these conventions is beyond the scope of this manual. However, you can find detailed information about it in Infineon's Titan reference manual and other generally available sources.

Before you begin

The component's description must be available as a subcircuit in a library in the project. You can make it available by adding it to the library manually or by importing a library containing the description. For more information, see [Working with libraries](#).

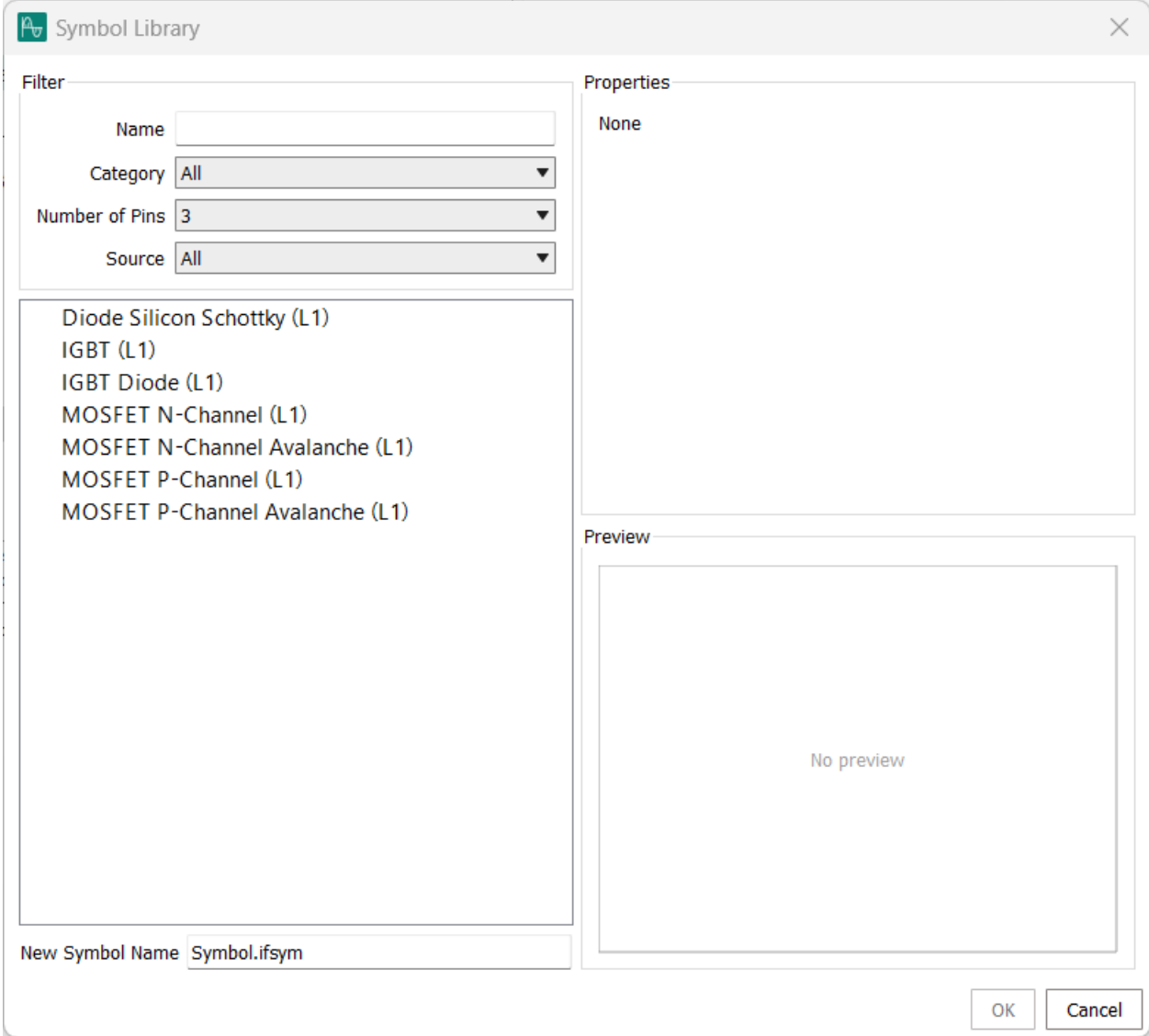
If you do not want to create the symbol during the process (as described below), the symbol must be contained in the library. For information on creating symbols, see [Working with symbols](#).

Steps

1. In the **Workspace** view, select the desired project and click the  button.
You can also choose **File > New Local Component** or right-click the project and choose **New Local Component** from the context menu.
The **New Device** dialog opens.
2. Enter the name that you want to give the local component and click **OK**.
The **Properties of Component** dialog opens, showing the **Basic** section.

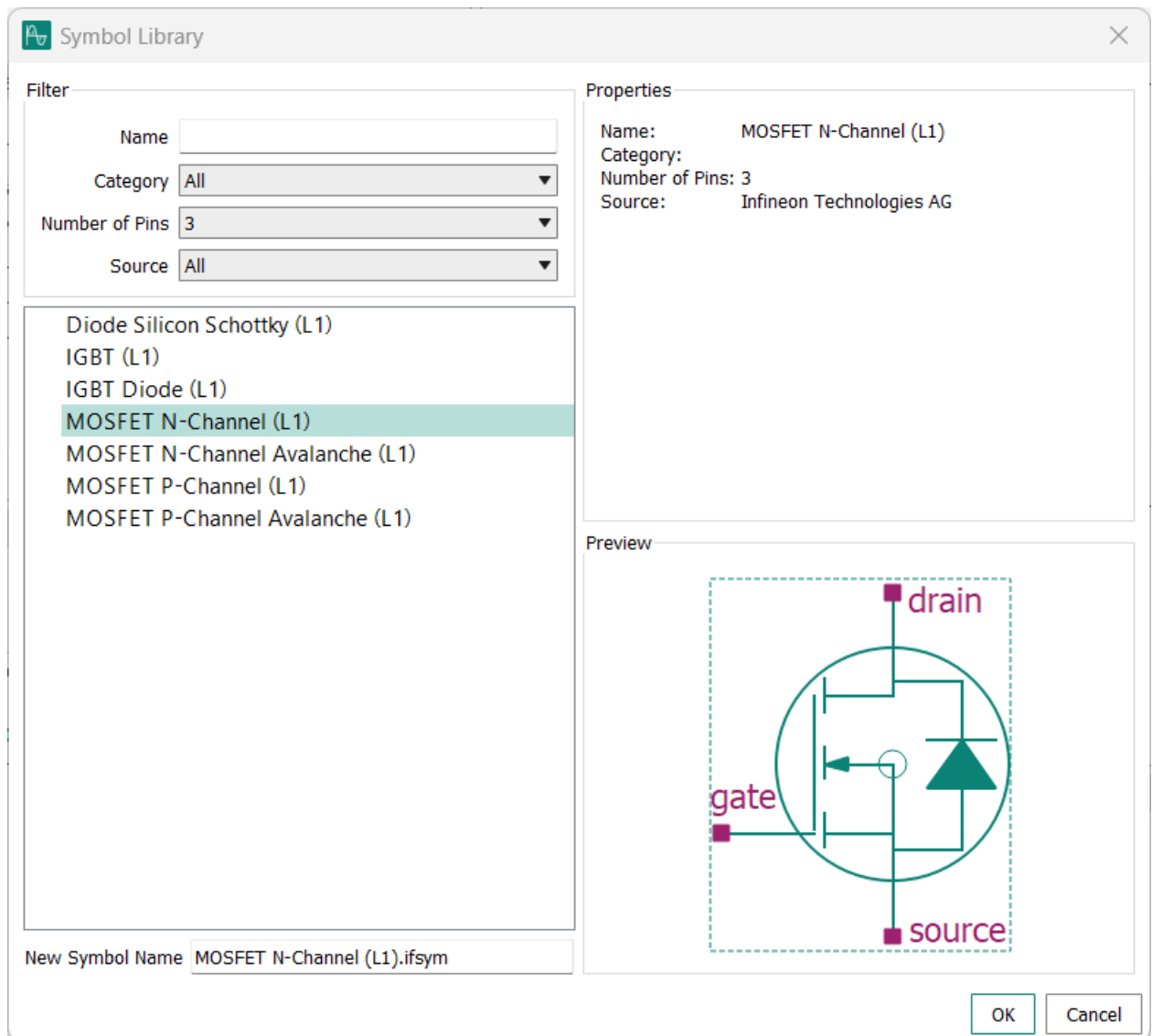
3 Using InfineonSpice

3. From the **Library** list, choose the library that contains the model that you want to use for the component.
4. From the **Model** list, choose the model.
5. Specify the symbol for the custom component.
 - If the symbol already exists in the project, choose it from the **Symbol** list. Note that you can choose only symbols with the same number of pins as the chosen model.
Continue with step [Step 15](#).
 - If you want to use a predefined symbol, continue with step [Step 6](#).
 - If the symbol does not exist at all yet, continue with step [Step 11](#).
6. Click **Default**.
The **Symbol Library** dialog opens.



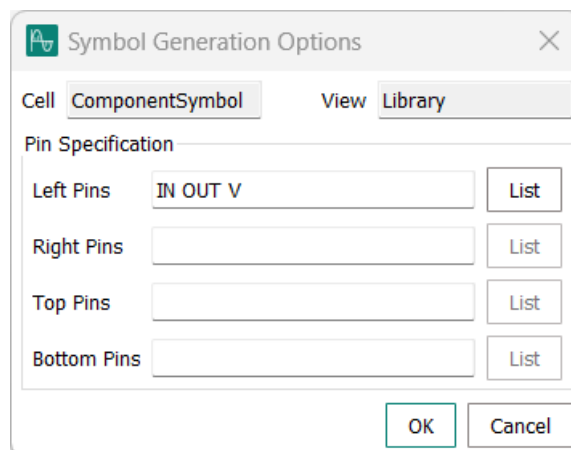
7. In the list of symbols, select the symbol that you want to use for the new component.
You can filter the list by entering part of the symbol name in the **Name** field, by choosing the desired category, the number of pins, or the source. If you specify multiple filter criteria, only symbols that match all of them are displayed.

3 Using InfineonSpice



8. Enter a name for the symbol in the **New Symbol Name** field.
Do not modify or delete the .ifsym filename extension.
9. Click **OK**.
The dialog closes and the symbol is added to the project.
10. Continue with step [Step 15](#).
11. Click **Generate**.
The **New Symbol** dialog opens.
12. Enter a name for the new symbol and click **OK**.
The **Symbol Generation Options** dialog opens.

3 Using InfineonSpice



All pin names defined by the model are listed in the **Left Pins** field, meaning that those pins will be positioned at the left edge of the generated symbol.

13. If you want to position pins at other edges of the symbol, remove their names from **Left Pins** and enter them into the desired fields.
14. Click **OK**.
The new symbol is created in the project and opened in a symbol editing tab in the background.
15. Click **OK** in the **Properties of Component** dialog to close the dialog.
The new custom component is created in the project.
16. If you have created the symbol during this procedure (and not used a predefined symbol), you can edit it now as desired in the tab that has automatically opened.
For information on editing symbols, see [Working with symbols](#).

Results

The new custom component is available in the project folder and can be used in schematics.

Note that you cannot edit the local component itself, but you can edit its symbol and its model separately using, respectively, the **Open in Symbol Editor** and **Open in Text Editor** context menu commands in the **Workspace** view.

3.6.1 Working with libraries

Libraries are text files containing SPICE netlist code. They can be edited with the built-in editor.

A library contains SPICE netlist code. It can, for example, be used to specify the behavior of local components or to load parts of a simulation.

Libraries are simple text files that can be edited with any text editor. InfineonSpice contains a custom editor that provides, in particular, syntax highlighting and searching with support for regular expressions. If you prefer an external editor, you can locate the file of a library that is part of a project by right-clicking it and choosing **Show in File Explorer** from the context menu.

Describing the SPICE code conventions and programming considerations is beyond the scope of this manual. However, you can find detailed information about it in Infineon's Titan reference manual and other generally available sources.

Importing and transferring libraries

You can import a library that you receive as a **.cir**, **.lib**, **.spi**, or **.tit** file by dragging it from the file system into the folder of a project in the **Workspace** view.

To make a library from one project available in another project, you can copy it by right-clicking it and choosing **Copy** from the context menu. This opens the **Copy File** dialog, where you can choose the target project and optionally give the copied library a new name. (Choosing the source project as the target and providing a new name effectively duplicates the library in its project.)

3 Using InfineonSpice

3.7 Setting up a simulation

Simulations are set up by specifying the desired parameters for them in the **Simulation Environment** view. Setting up a simulation includes specifying the desired result categories, simulation parameters, and output options. While it is possible to run a simulation with default settings, getting the information required in a particular stage of the development usually requires an appropriate setup for the simulation. Specifying a setup includes entering numerous values. Options for doing so are described in [Values and expressions](#).

Before you begin

The circuit that you want to simulate must be available in the project as a schematic.

Steps

1. Open the schematic to be simulated.
The tab displaying it must be the frontmost tab.
2. In the **Simulation Environment** view's **Analyses** tab, check the areas that the simulation should analyze.
For each of the areas – transients, operating points, DC, AC – a subtab is displayed where you can specify basic settings for the analysis.
These settings are explained in detail in [Simulation Environment view](#).
You can set up multiple analyses at the same time. In that case, running the simulation opens a separate Waveform Viewer window for each specified type of analysis.
Note that an AC analysis requires the power source of the schematic to have an ACM setting.
3. Specify the desired accuracy and temperature at the bottom of the **Analyses** tab.
Note that higher accuracy slows down the simulation. What SPICE settings correspond to each choice is displayed when you hover the mouse over the corresponding radio button.
4. In the **Parameters** tab, specify the settings for any parameters that you have defined.
Any values that depend on parameters (see [Values and expressions](#)) are calculated for the simulation using the parameter settings in this tab.
For example, when a value is specified as $2 * \text{baseRate} + 1$ and baseRate is set to 5, the value is 11 during this simulation. If you set baseRate to 5.5, the value calculated from it is 12.
Note that the value assigned to a parameter can be static values or expressions (see [Values and expressions](#)) and can include other parameters.
5. In the **Plot Setup** tab, specify which voltages and currents are recorded for evaluation.

All

Every voltage or current, including in subcircuits

Top-level

Every voltage or current in the schematic itself, but not those in subcircuits

Signal only

Only voltages or currents on the specified signal

Recording more voltages or currents gives you more flexibility in the evaluation of the simulation, but the simulation takes longer to run.

You can define additional signals in the schematic using the Select Output tools  and  (standard and differential, respectively). You can also define expressions to plot based on signals by double-clicking the **New expression** entry, which opens the **Expression** dialog.

The AC analysis requires special signal setups, as described in [Setting up an AC analysis](#).

3 Using InfineonSpice

If you set up multiple analyses, you can specify the same signal for more than one of them. To do so, add the signal once for each analysis and select the desired type of analysis for each of them in the **Analysis** column.

6. From the tab's **Plot Mode** list, choose how the simulation results are to be displayed.
This setting determines just the initial display of the results. You can always add, remove, and rearrange plots during evaluation. For details, see [Working with plots](#).
7. If external libraries for subcircuits in the schematic are required, specify them in the **Model Libraries** tab.
Enter the path of each library file, either as an absolute path or a relative path.
8. In the **Settings** tab, specify how floating nodes are to be handled.
By default, a floating node is handled as if it were connected to ground through a 1 G Ω resistor.
9. If you want to add custom netlist commands to the simulation, enter them in the **Additional Netlist Commands** tab.
Any netlist code you enter here is appended to the end of the netlist that InfineonSpice creates for the simulation.

Results

The simulation is now ready to run.

What to do next

You can run the simulation, as described in [Running a simulation](#).

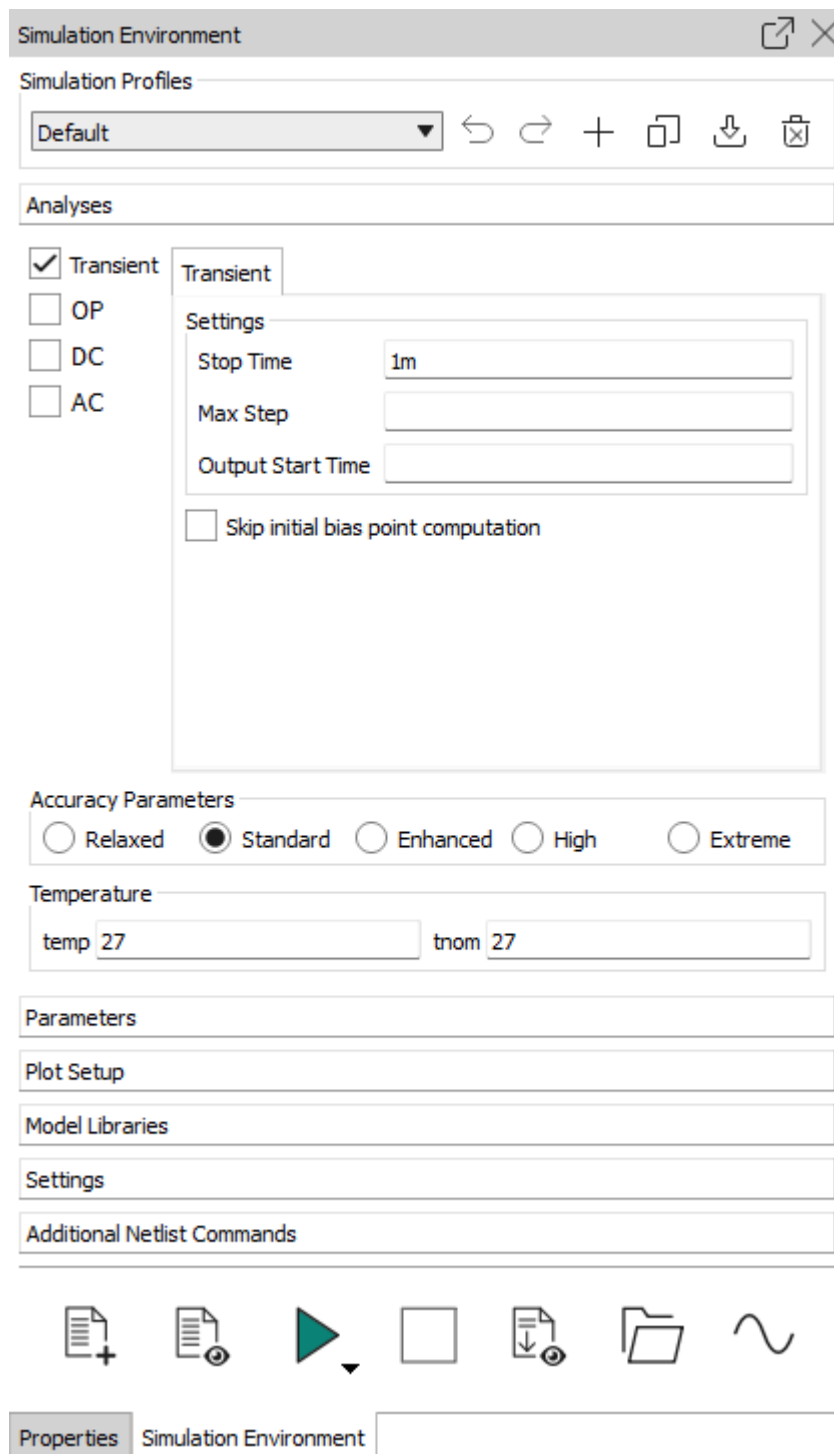
3.7.1 Defining simulation profiles

Simulation profiles let you store settings that you commonly use. They can be recalled simply by choosing them from the **Simulation Profiles** menu.

Creating a simulation profile for the low pass filter

1. Define the profile in the **Simulation Environment**, which is initially displayed on the right side of the design environment
2. If you switched to the **Properties** tab while drawing the schematic, you have to switch back to it by clicking on **Simulation Environment** on the bottom right of the InfineonSpice design environment.

3 Using InfineonSpice



The **Simulation Environment** dialog box is shown with the following sections and controls:

- Simulation Profiles:** A dropdown menu set to "Default", with buttons for undo, redo, add, copy, import, and delete.
- Analyses:** A list of analysis types:
 - ☒ Transient: The **Transient** sub-tab is active, showing settings for Stop Time (1m), Max Step, Output Start Time, and a checkbox for "Skip initial bias point computation".
 - ☐ OP
 - ☐ DC
 - ☐ AC
- Accuracy Parameters:** Radio buttons for Relaxed, **Standard** (selected), Enhanced, High, and Extreme.
- Temperature:** Input fields for "temp" (27) and "tnom" (27).
- Parameters**
- Plot Setup**
- Model Libraries**
- Settings**
- Additional Netlist Commands**

At the bottom, there is a toolbar with icons for adding a new profile, deleting a profile, running simulation, pausing simulation, saving, opening a file, and a waveform icon. Below the toolbar are tabs for **Properties** and **Simulation Environment**.

At the top, the **Simulation Environment** consists of the following items:

- Drop down-menu which allows to switch between different simulation profiles
- Undo/Redo to revert changes in simulation profiles
- Button to add new simulation profiles
- Button to copy simulation profiles
- Button to import a simulation profile from another schematic
- Button to delete the currently selected simulation profile

At the center, the **Simulation Environment** has different sections which can be used to define the following:

- Different circuit analysis types include Transient, OP, DC

3 Using InfineonSpice

- The circuit's design parameters, for example supply voltages or transistor widths
- The plot setup of the analysis, for example specific node voltages and device currents to be plotted
- File paths to model libraries which contain device models used in the circuit
- Simulator settings, for example accuracy parameters
- Additional Netlist commands, that is, user-defined inputs in TITAN syntax

Defining simulator outputs

The simulator outputs, that is, the signals to be plotted by the waveform viewer after the successful termination of a simulation, are defined in the **Plot Setup** section of the **Simulation Environment**. By default, only **Top-Level** voltages and currents of the circuit are stored in the results data base. Signals deeper down the circuit hierarchy are not saved.

- Option **All** allows to store all voltages and currents of the circuit, which however increases the amount of data generated by TITAN
- Option **Signals only** causes TITAN to save only the signals specified in the **Signals** pane, minimizing the disk space required for saving the outputs

3.7.2 Setting up a transient analysis

To set up a transient analysis, only the duration of the analysis has to be specified. Optionally additional settings can be made.

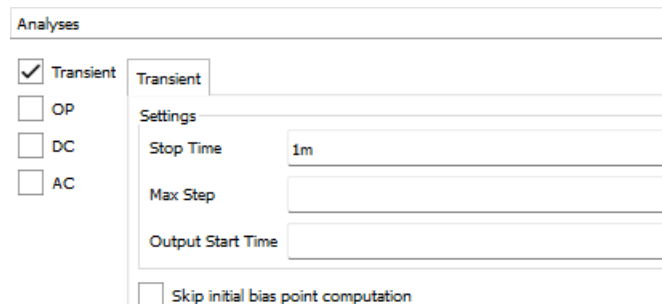
Before you begin

Set up the simulation as described in [Setting up a simulation](#).

Set up a transient analysis when you want to analyze the state of a device in rest.

Steps

1. In the **Analyses** tab of the **Simulation Environment** view, check **Transient** if it is not already checked.



You can check additional analyses (such as OP or DC). If you do so, an additional Waveform Viewer window will open for each analysis when the simulation is run.

2. In the **Stop Time** field of the **Transient** tab, specify how much time should be simulated.
For information on the number format, see [Values and expressions](#).
3. Optionally, specify the maximum step size as an interval (for example, 5u).
The step size of the simulation is dynamically determined during the simulation; steps may have different sizes. By specifying a maximum step size, you ensure that no individual step is larger than this interval.
Shorter maximum step sizes increase the resolution of the simulation but also the running time and the size of the result files.
4. Optionally, specify a starting time for the result data in the **Output Start Time** field.

3 Using InfineonSpice

This does not influence the start of the simulation (which always starts at 0 s), but the simulation results will only be displayed and saved to disk after this point. This is useful, for example, if you are interested in the behavior of the device only after the settling time.

Results

The transient analysis is now set up, and you can run the simulation as described in [Running a simulation](#). Evaluating the simulation results is described in [Evaluating simulations](#).

3.7.3 Setting up an operating point analysis

No additional setup is necessary for an operating point analysis. If desired, additional in time can be specified.

Before you begin

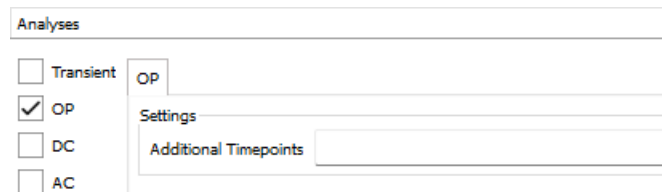
Set up the simulation as described in [Setting up a simulation](#).

Set up a operating point analysis .

Note that a transient analysis automatically performs an operating point analysis at 0 s.

Steps

1. In the **Analyses** tab of the **Simulation Environment** view, check **OP** if it is not already checked.



You can check additional analyses (such as DC or AC). If you do so, an additional Waveform Viewer window will open for each analysis when the simulation is run.

2. Optionally, specify additional points in time at which an operating system is performed.
InfineonSpice always performs an operating point analysis at 0 s. You can specify additional points in time.
Specify each point as an interval from 0 s, in the number format described in [Values and expressions](#).
Separate individual numbers with spaces.
For example, to specify operating point analyses at 0.1 s and 2 s, enter 100m 2.

Results

The operating point analysis is now set up, and you can run the simulation as described in [Running a simulation](#). Evaluating the simulation results is described in [Evaluating simulations](#).

3.7.4 Setting up a DC analysis

To set up a DC analysis, specify the sweep value and the sweep range.

Before you begin

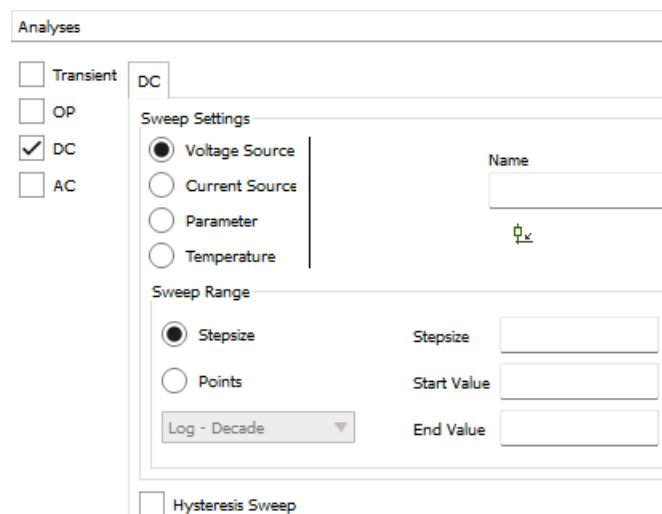
Set up the simulation as described in [Setting up a simulation](#).

Set up a DC analysis when you want to analyze the behavior of a device when a particular property is varied.

Steps

1. In the **Analyses** tab of the **Simulation Environment** view, check **DC** if it is not already checked.

3 Using InfineonSpice



You can check additional analyses (such as OP or AC). If you do so, an additional Waveform Viewer window will open for each analysis when the simulation is run.

2. Choose the desired sweep setting.

- **Voltage Source:** Enter the name of the voltage source in the **Name** field, or click the Select Output tool  below it and click the voltage source in the schematic.
- **Current Source:** Enter the name of the current source in the **Name** field, or click the Select Output tool  below it and click the current source in the schematic.
- **Parameter:** Enter the name of the parameter in the **Name** field. If the parameter has been defined in the schematic itself, leave the **Subcircuit** field empty. If the parameter has been defined in a subcircuit, enter its name into the field.
- **Temperature:** No additional options are available for this sweep setting.

3. Specify the type of sweep range.

- To specify range by the step intervals, choose **Stepsize** and enter the desired interval between two steps in the **Stepsize** field.
For information on the number format, see [Values and expressions](#).
For example, when a voltage source is swept, entering 250m sweeps the range in steps of 0.25 V.
- To specify range by the number of steps, choose **Points** and enter the desired number of simulated points in time in the **# Points** field and choose linear or logarithmic spacing.
Note that the number of steps includes both the beginning and end of the range, so that entering, for example, 6 results in a sweep with five steps. If you choose **Linear**, the steps are all equally-sized. If you choose **Log - Decade** or **Log - Octave**, each step is logarithmically larger than the one before.

4. Enter the beginning and end of the range in the **Start Value** and **End Value** fields.

For information on the number format, see [Values and expressions](#).

5. Optionally, check the **Hysteresis Sweep** checkbox to add a second sweep after the first one. This second sweep has the same parameters as the first one but begins with the specified end value and ends with the start value.

Results

The DC analysis is now set up, and you can run the simulation as described in [Running a simulation](#). Evaluating the simulation results is described in [Evaluating simulations](#).

3 Using InfineonSpice

3.7.5 Setting up an AC analysis

To set up an AC analysis, the frequency range must be specified and AC-specific parameters for voltage and power sources be set. For signals, the desired component to display must be specified.

Before you begin

Set up the simulation as described in [Setting up a simulation](#), but do not specify any signals yet.

Set up an AC analysis when you want to analyze the frequency response of a device.

Steps

1. Double-click the voltage source of the schematic.
The **Properties** dialog opens.
2. Set the ACM parameter and, if desired, the ACP parameter.

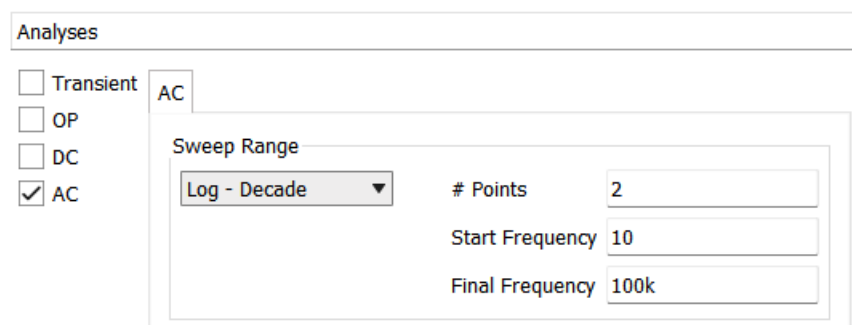
ACM

The AC magnitude (acmag). When performing AC analyzes, this parameter is used instead of the standard VA setting of the voltage source.

ACP

The AC phase (acphase). An optional setting specifying the starting phase of the voltage source's output.

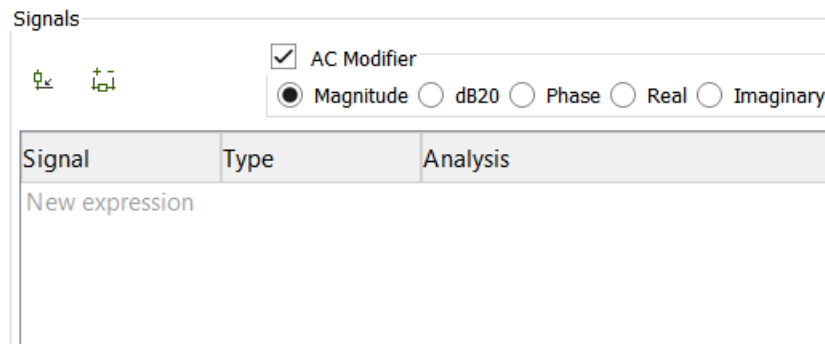
3. Repeat these steps for any other voltage and current sources.
4. In the **Analyses** tab of the **Simulation Environment** view, check **AC** if it is not already checked.



You can check additional analyses (such as OP or DC). If you do so, an additional Waveform Viewer window will open for each analysis when the simulation is run.

5. Specify the sweep range for the frequency in the **AC** tab.
Set the desired type of sweep: log based on decades, log based on octaves, or linear. This determines the distribution of the points.
Set the number of points. The simulation will be performed at each point. For logarithmic ranges, each decade or octave section has the specified number of points. For linear ranges, the number of points is distributed across the entire sweep range.
Set the desired start and end frequencies.
6. In the **Plot Setup** tab, specify the desired signals in the **Signals** section.

3 Using InfineonSpice



- a. Check the **AC Modifier** option.
- b. Specify the desired component that you want to graph: magnitude, attenuation (**db20**), phase, real component, or imaginary component.
This setting affects just the initial display in the Waveform Viewer. You can add the other modifiers as separate traces without having to rerun the simulation.
Note that the real and imaginary components of the named voltages and currents of the schematic are automatically added to the analysis and do not need to be specified as signals. However, other aspects of these values, such as attenuation or phase, are not automatically added. If you want to have them included in the initial plot, you must add them as signals. Alternatively, you can add them when reviewing the simulation results, as described in [Evaluating AC analyses](#).
- c. Click the appropriate selection tool in the upper left corner of the section and use it to select the desired signal in the schematic that you want to include in the simulation.
The signal is added to the signal list.
- d. In the **Analysis** column of the newly added signal, choose **AC**.
- e. Repeat these steps for any other signals you want to add to the analysis.

Results

The AC analysis is now set up, and you can run the simulation as described in [Running a simulation](#). For evaluating the simulation results, everything in [Evaluating simulations](#) applies. Specific considerations for AC analyses are described in [Evaluating AC analyses](#).

3.8 Running a simulation

When a schematic has been set up, it can be simulated. This automatically creates the required netlist, saves the simulation results, and displays them for analysis in the Waveform Viewer.

Simulating a schematic lets you view the currents and voltages at specified points of interest over time. The simulation is controlled using the toolbar at the bottom of the **Simulation Environment** view



Note that rerunning a simulation overwrites any previous results from simulating the same schematic. If you want to be able to access the results of an earlier simulation, you must first make a copy of them in the file system. The results are stored in a subfolder of the project folder with the name `sim-schematicName-Default`. If you make a copy of that folder, you can later open the `.csdf` file in it using the **File > Open** command in Waveform Viewer.

Before you begin

The schematic must be completed, as described in [Working with schematics](#). It must be displayed in the frontmost tab.

The simulation must be set up, as described in [Setting up a simulation](#).

3 Using InfineonSpice

Steps

1. If you want to review the netlist before running the schematic, click the Create Netlist button .
The netlist is created and shown in a new tab.
You can edit the netlist in this tab and save the changes for use in the simulation. However, we recommend that you do not do so and instead make any required adjustments in the schematic. If you do edit the netlist, do so only with the utmost caution, because any editing errors can invalidate the simulation results.
2. Click either  or  to run the simulation.
You can switch between the two buttons by clicking the triangle in their lower-right corner and choosing the desired button from the menu that opens.

Normally, you run the simulation using the  button. This recreates the netlist and immediately runs the simulation. However, if you have already generated the netlist and edited it (as described in step [Step 1](#)), this overwrites your changes. In that case, use the  button which runs the simulation using the existing netlist, without recreating it first.
If a simulation runs too long, you can abort it by clicking . In that case, no simulation results are created and the Waveform Viewer is not opened.

Results

The simulation is run with the specified settings. Diagnostic output is written to the tabs of the **Output** view. The Waveform Viewer is opened, displaying the simulation results.

What to do next

You can review the simulation results in the Waveform Viewer, as described in [Evaluating simulations](#). If the Waveform Viewer does not open automatically, you can open it manually by clicking .

You can review the log messages generated during the simulation by clicking .

You can access the files generated by the simulation by clicking .

3.8.1 Output files of a simulation

Running a simulation creates a range of files in the output folder.

The created files are:

Proprietary result file (.apb)

APB folders contain the results of a simulation in a proprietary format. They are collections of individual files that store the various aspects of the result data (among them .apb files).

The content of an APB folder is similar to the content of a CSDF file, but APB folders cannot be used for data exchange with other tools.

Saving results in Waveform Viewer creates APB folders.

CSDF result file (.csdf)

CSDF files contain the simulation results in a text-based format that enables data exchange with other tools.

Output log (.lis)

3 Using InfineonSpice

The log file of the simulation process.

Input processor log (.msg)

The log file of the input processor of the simulation engine.

Netlist (.net)

The netlist that was used for the simulation.

Titan-specific netlist (.tit)

A version of the netlist that includes settings specific to Titan, the simulation engine of InfineonSpice.

Other files

Other output files are intermediate files that are created when a simulation is run. These files contain no independently usable data. File types created include, among others:

analogBlocks, .cdf, .dir .dp, .lock, .ppw.

Some of these files are not shown in the main InfineonSpice window but can be accessed by right-clicking the output folder in the **Workspace** view and choosing **Show in File Explorer**.

3.8.2 Running InfineonSpice from the command line

By setting up a batch script, it is possible to create the result files of the simulation of a predetermined schematic from the command line.

Before you begin

Create the schematic, set up the simulation as desired, and run it or otherwise create a netlist from it.

Using this technique, you can automate the simulation of a schematic to quickly generate the output files of the simulation without having to start InfineonSpice and run the simulation manually in the application.

Steps

1. Put the netlist file (.net) that you want to simulate in a convenient location.
The output of the simulation will be created in the same directory.
2. Change to the titan_core directory in the InfineonSpice installation directory.
By default, InfineonSpice is installed in C:\Users\username\AppData\Local\Programs\Infineon\InfineonSpice\version-number\. For example, the titan_core folder might be found at C:\Users\JaneDoe\AppData\Local\Programs\Infineon\InfineonSpice\2025.03\titan_core.
Instead of changing the active directory, you can also execute titan.bat by prefixing its path, as described in the following step.
3. Simulate your netlist by entering titan.bat path-to-netlist.
For example: titan.bat C:\\Users\\JaneDoe\\Documents\\Simulation\\MyNetList.net.
If you did not change the active directory as described in the previous step, you can execute titan.bat by prefixing its path. For example: C:\\Users\\JaneDoe\\AppData\\Local\\Programs\\Infineon\\InfineonSpice\\2025.03\\titan_core\\titan.bat C:\\Users\\JaneDoe\\Documents\\Simulation\\MyNetList.net
Note that the name of the netlist file must not contain spaces.

Results

The simulation is executed and the output files are created in the same directory as the simulated netlist file. For an overview of the generated files, see [Output files of a simulation](#).

The created files have the same names as the netlist file (except for the extensions).

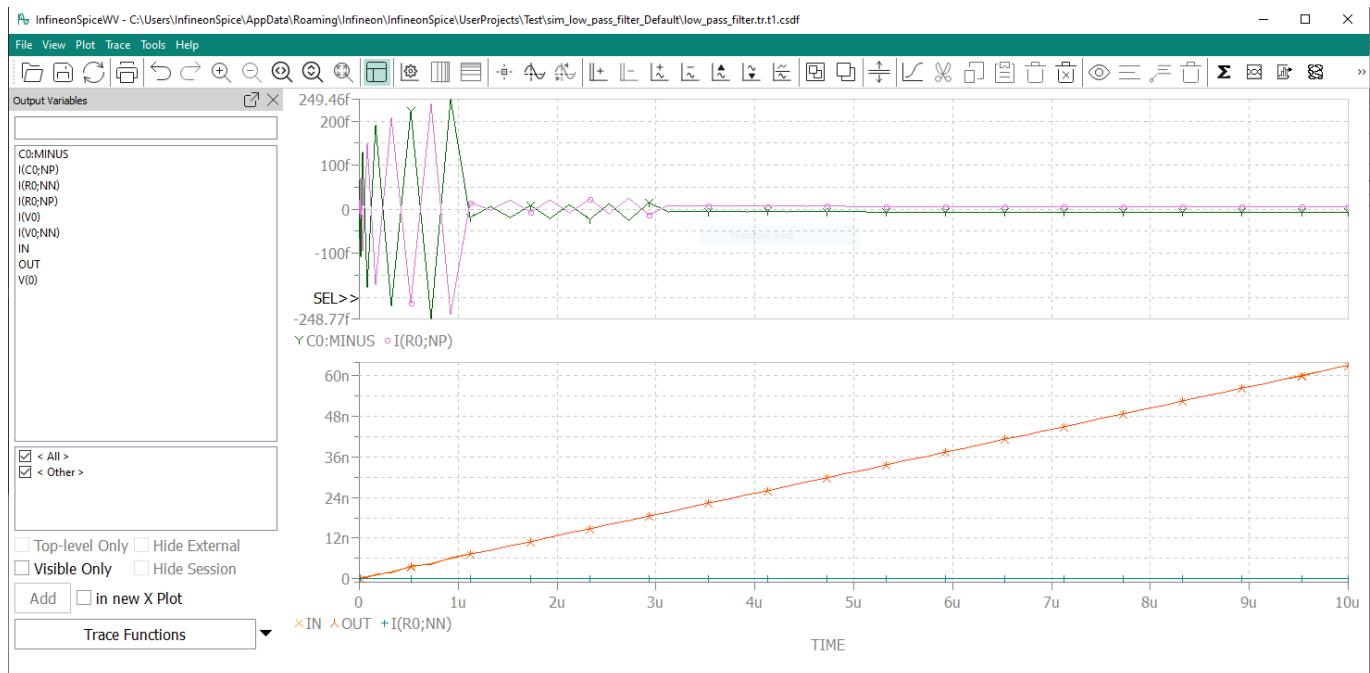
3 Using InfineonSpice

3.9 Evaluating simulations

Simulations are evaluated in InfineonSpice's Waveform Viewer module by examining the plots from the simulated values.

To evaluate a simulation, you must have run it as described in [Chapter 3.8](#). Doing so creates result files that Waveform Viewer can display.

Waveform Viewer has its own window outside the main InfineonSpice window:



It contains a menu bar and toolbar across the top, the **Output Variables** view at the left (although it can be hidden) and the plots in the main part of the window. A plot is a rectangular diagram containing one or more traces (series of simulation result values), axes, and optionally annotations in the form of labels. The Waveform Viewer window can contain multiple plots, which are stacked vertically.

Note that, in results of AC analyses, each output variable is represented by two entries, one for the real component and one for the imaginary component.

Working with results

Usually, Waveform Viewer is automatically opened when the simulation is complete, displaying the simulation's results. However, you can load other saved results using the **File > Open** command. Simulation results are stored in subfolders of the project folder with the name *sim-schematicName-Default* in files whose names end in *.csdf*.

You can also open the saved waveform files in InfineonSpice by double-clicking the files in the simulation's result folder in the workspace. To open two files in one Waveform Viewer window, drag one file on top of the other in InfineonSpice.

If Waveform Viewer displays results and the result files on disk change, you can update the display using the **File > Reload** command.

Working with plots

When a simulation has been completed, Waveform Viewer displays the results in one plot. You can create additional plots, remove existing plots, and rearrange plots, as described in [Chapter 3.9.1](#).

The axes in plots can be dimensioned automatically or manually. It is also possible to add and remove y-axes when a plot contain multiple traces. See [Chapter 3.9.3](#) for details.

Plots can also be annotated to emphasize aspects of interest or improve their comprehensibility, as described in [Chapter 3.9.4](#).

3 Using InfineonSpice

Working with traces

Each plot contains one or more graphical representations of series of simulated values, so-called traces. You can add and remove traces, transfer traces between plots, and modify their appearance. For details, see [Chapter 3.9.2](#).

3.9.1 Working with plots

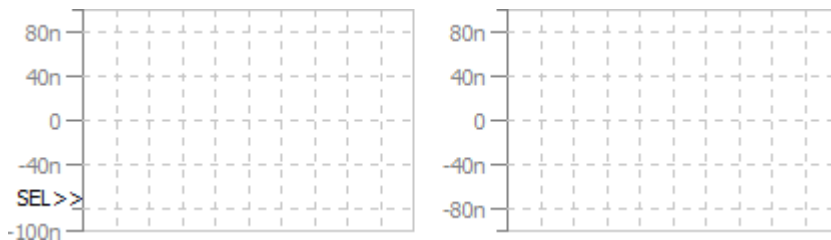
Plots are diagrams in Waveform Viewer that display traces. You can add, remove, and rearrange them. You can also zoom in and out in various ways

Plots are used to [evaluate a simulation](#).

Selecting plots

Some operations with plots require you to select a plot first. To select a plot, simply click it. The currently selected plot is indicated by the text mark SEL>> on the left-hand side.

The first plot is selected, the second one is not:



Adding and removing plots

To add another plot in the Waveform Viewer window, choose **Plot > Add Plot** or click  in the toolbar. In both cases, a new empty plot is added at the top of the window. See [Working with traces](#) for information on adding traces to the new plot.

To remove an existing plot, select it and choose **Plot > Delete Plot** or click  in the toolbar. A plot is also removed automatically when you remove the last trace from it.

To remove all plots except one while keeping the displayed traces, choose **Plot > Combine Traces** or click  in the toolbar. All traces are collected in a single plot, and the other plots are removed.

Rearranging and resizing plots

Multiple plots displayed in Waveform Viewer are arranged vertically. To move a plot higher, select it and then choose **Plot > Move Plot Up** or click  in the toolbar. To move a selected plot lower, choose **Plot > Move Plot Down** or click .

You can also modify the height of a plot. To do so, choose **Plot > Resize Plots Manually** or click  to select the resizing tool. Then click the upper or lower border of the plot and drag it up or down. Adjacent plots are resized automatically to fill the available space. If only a single plot is shown in the Waveform Viewer window, it is not possible to resize that plot. (However, you can resize the window, which resizes the plot accordingly.)

To return all plots to their natural sizes, choose **Plot > Adjust Size > Auto**.

Zooming

You can zoom plots to enlarge or reduce the content in a variety of ways:

- To make the plot content larger, select the desired plot and choose **View > Zoom > Zoom Increase** or click  in the toolbar.

You can also make sure that no tool is selected (the standard arrow mouse pointer is displayed) and drag a rectangle around a section of a plot. The plot is zoom in on that area.

3 Using InfineonSpice

- To make the plot content smaller, select the desired plot and choose **View > Zoom > Zoom Decrease** or click  in the toolbar.
- To see the entire content of a plot, select it and choose **View > Zoom > Zoom All** or click  in the toolbar.
To see the entire width while leaving the height zoom unchanged, click . To see the entire height without changing the width zoom, click .
- When you have changed the zoom level in any way, you can return to the previous zoom level by selecting the plot and choosing **View > Zoom > Zoom Back** or clicking  in the toolbar. If you have gone back in this manner, you can go forward in the history of zoom levels again by choosing **View > Zoom > Zoom Forward** or clicking .

By default, all plots have synchronized x-axes so that they all show the same time period from the simulation, and zooming one plot changes the x-axes of the other plots accordingly. If that is not desired, you can choose **Plot > Unsynchronize X Axis** or click  in the toolbar.

Working with plot content

For information on modifying the contents of plots, see:

- [Working with traces](#)
- [Working with axes](#)
- [Working with annotations](#)

3.9.2 Working with traces

Traces are data series depicted in plots. The data can be simulation values or functions based on simulation values. You can add, remove, and rearrange traces.

Traces are used to [evaluate a simulation](#).

Selecting traces

You select a trace by clicking it or clicking its entry in the legend below the plot. You can select multiple traces by shift-clicking or control-clicking them. (Both methods have the same effect.)

Adding and removing traces

When the Waveform Viewer first opens, it contains traces for the signals you have specified when setting up the simulation. You can add more traces or remove traces that you do not need.

To remove a trace, click it in the plot to select it and press the Del key or click the  toolbar button. To delete all traces in the [selected plot](#), click  in the toolbar. If you have accidentally deleted one or more traces, you can choose **Trace > Undelete** to undo this action.

You can add different types of traces to the [selected plot](#):

- To add a signal, double-click it in the **Output Variables** view, select it and click the **Add** button at the bottom of the view, or drag it from that view to the desired plot. The options below the list of signals let you filter it. (If the view is not displayed, click  in the toolbar to show it.)

You can check the **In new X plot** option at the bottom of the view before double-clicking the signal or clicking **Add** to display the trace in a new plot instead of the selected plot. That new plot has an independent (unsynchronized) x-axis.

You can also add a trace using the **Trace > Add Trace** command or clicking  in the toolbar. This opens the **Add Trace(s)** dialog where you can select one or – by control- or shift-clicking – multiple signals or variables that are added when you click **OK**.

Choosing **Trace > Add All** adds all signals and variables from the **Output Variables** view to the selected plot.

3 Using InfineonSpice

If the plot to which you add a trace has multiple y-axes, the traces are added to the **selected y-axis**.

- To add a trace based on an equation, click the **Trace Functions** button at the bottom of the **Output Variables** view. This opens the **Evaluate Goal Function** dialog, in which you can enter the desired equation. Click **OK** to display the equation results as a trace in the selected plot. If you are adding traces using the **Add Trace(s)** dialog, you can click **Eval Goal Function** instead.

Similar to adding signal traces (described above), you can choose **in new X plot** from the list in the bottom right corner of the dialog to place the trace in a new plot with an independent x-axis. In the **Add Trace(s)** dialog, you get the same result by checking **Place in New X Plot**.

Any equations you create are saved as variables in the list in the **Output Variables** view and can be thereafter placed in the same way as signals.

Moving traces

You can move traces between plots:

- To move a trace to a different plot, select it and drag it to the desired plot. When you drop it, the y-axes of the target plot is automatically adapted: If necessary, the range of the axis is increased to accommodate the value range of the trace.
To duplicate the trace instead of moving it, hold down the Ctrl key while dragging it to another plot.
- You can also move a trace by copying and pasting it, using the **Cut Trace**, **Copy Trace**, and **Paste Trace** commands from the **Trace** menu or the equivalent toolbar buttons , , and . Note that pasted traces are inserted into the **selected plot**.
- To collect all traces in one plot, choose **Plot > Combine Traces** or click  in the toolbar. All traces are collected in a single plot, and the other plots are removed.
- To display each trace in a plot of its own, choose **Plot > Scatter Traces and Zoom** or click  in the toolbar. This creates for each trace a plot with auto-sized y-axis. If you prefer instead to use the y-axis from the original plot in each generated plot, choose **Plot > Scatter Traces and Zoom** instead. Note that both commands affect all plots in the Waveform Viewer window, not just the selected one.

Modifying the appearance of traces

You can change both the color of the trace and the shape of the marks indicating the simulated or calculated values.

By default, both colors and shapes are assigned automatically. If you want to use different appearances, select the trace and choose **Trace > Properties** to open the **Trace Properties** dialog. In that dialog you can click **Custom Color** to assign a different color to the trace by clicking on the color swatch or **Custom Mark** to assign a different shape by clicking on the shape symbol.

The chosen color and shape is assigned to the trace when you click OK. If you first check the **Apply to all the same traces** option, all other traces that display the same signal or variable are displayed with the same color and shape.

If you want to revert the color or mark shape of an individual trace to the defaults, open the **Trace Properties** dialog and choose **Current Color Scheme**, **Current Mark Scheme**, or both. If you want to reset the colors or mark shapes of all traces in all plots, choose **Traces > Attributes > Reset All Trace Color**, **Traces > Attributes > Reset All Trace Mark**, or both.

You can toggle the display of the connecting lines in the traces by choosing **Traces > Trace Visibility > Hide Lines**. To toggle the display of the marks, choose **Traces > Trace Visibility > Hide Marks**. Note that either the lines or the marks must be visible. Therefore, if the lines are hidden and you choose **Hide Marks**, the lines are automatically displayed, and vice versa.

Renaming traces

It is not possible to rename a trace, but you can give it a display name under which it is listed in the **Output Variables** view. The same display name is also used for labeling traces in plot legends.

To assign a display name, choose **Trace > Aliases** and enter the desired name in the right-hand column beside the signal or variable name.

3.9.3 Working with axes

Each plot has a x-axis and one or more y-axes which indicate the values of the plotted data points.

The x-axis shows the time in the simulation when each value was calculated, and the y-axes show the size of the values. Each plot has one x-axis but can have multiple y-axes for multiple traces.

X-axis

The x-axis displays the simulation time. By default, there is one x-axis for all plots together. Zooming or scrolling a plot zooms and scrolls all other plots in the same way, ensuring that you can always readily compare the values in all traces.

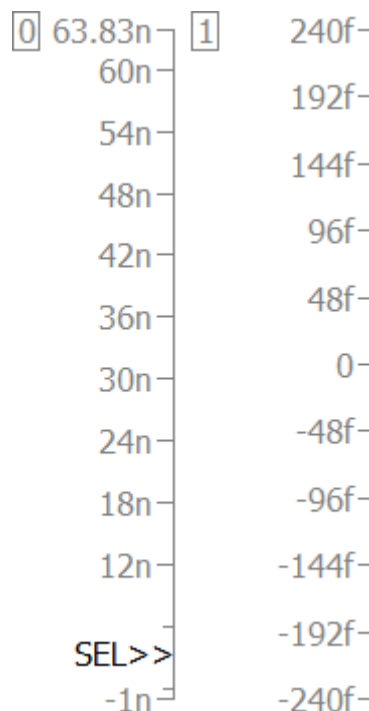
Sometimes, this synchronization of the x-axis may not be desired. In such cases, you can make the x-axis of each plot independent from the others by choosing **Plot > Unsynchronize X-Axis** or clicking  in the toolbar. If you want to assign a fixed custom value range to a x-axis, you can select a plot that belongs to that x-axis and choose **Tools > Axis Settings** or click  in the toolbar. This opens the **Axis Settings** dialog, in which you can specify the end points and step width of the axis, choose whether it is linear or logarithmic, and give the axis a custom label.

If you want to switch the axis between linear and logarithmic display, you can do so in the **Axis Settings** dialog, as described above, or click  in the toolbar. Note that an axis can only be switched to logarithmic display if it does not contain zero.

Selecting a y-axis

Some operations you can perform with y-axes or traces require you to select the desired y-axis first. To do so, simply click it. The currently selected y-axis is indicated by the text mark SEL>>. (The plot containing the axis is simultaneously selected for any plot-related operations, as described in [Working with plots.](#))

The first axis is selected, the second one is not:



Multiple y-axes

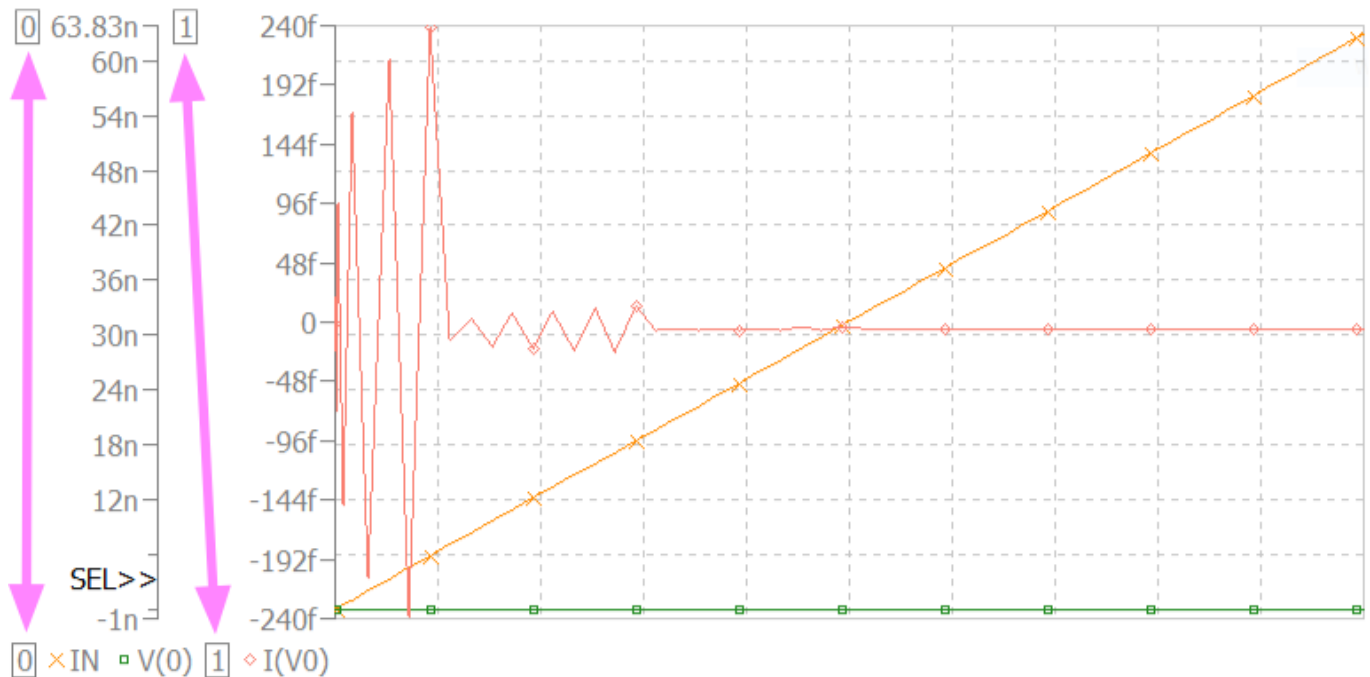
When a plot is created, it has one y-axis. By default, the y-axis is sized automatically so that it covers the full range of the values in all traces in the plot.

3 Using InfineonSpice

One y-axis for all traces of a plot can mean that some traces are not very readable. For example, if one trace has an amplitude of 100 mA and the other just 3 mA, the second one will look very much like a horizontal line. To be able to meaningfully compare traces with greatly differing amplitudes, you can add separate y-axes for them.

To do so, [select the plot](#) and choose **Plot > Add Y-Axis** or click  in the toolbar. The axis has default settings. If you next add a trace to the plot (see [Adding and removing traces](#) for details) or move one there (see [Moving traces](#)), the trace is assigned to that axis and the axis is automatically scaled for it.

If a plot contains multiple y-axes, they are numbered consecutively, starting with zero. The trace listing below the plot is grouped by the axes to which each trace belongs. In this illustration, IN and V(O) are displayed according to axis 0, and I(V0) is displayed according to axis 1:



To assign a trace that belongs to one axis to another axis, select the target axis and drag the trace label from below the plot into the plot. In the example above, dragging the I(V0) label from below the plot into the plot would assign that trace to the 0 axis.

To delete a y-axis from a plot, select it and choose **Plot > Delete Y-Axis** or click  in the toolbar. Note that deleting an axis also deletes all traces that are assigned to it. If you want to keep those traces, assign them to another y-axis first, as described above.

Custom y-axes

By default, a y-axis is automatically sized to accommodate the value range of the traces that belong to it, and it is resized when a new trace with a larger value range is added.

If you want to keep an axis at its current size even when more traces are added later, you can deactivate this automatic accommodation by unchecking the **Plot > Auto-Scale Y-Axis** command.

If you want to assign a fixed custom value range to a y-axis, you can select it and choose **Tools > Axis Settings** or click  in the toolbar. This opens the **Axis Settings** dialog, in which you can specify the end points and step width of the axis, choose whether it is linear or logarithmic, and give the axis a custom label.

If you want to switch the axis between linear and logarithmic display, you can do so in the **Axis Settings** dialog, as described above, or click  in the toolbar. Note that an axis can only be switched to logarithmic display if it does not contain zero.

3 Using InfineonSpice

3.9.4 Working with annotations

Plots created in Waveform Viewer can be annotated with labels to indicate features of interest or to otherwise convey information to human readers.

Note that you can see and work with labels only if their display is switched on, which is indicated by the

activated state of the Show/Hide Labels toolbar button: . If the button is not activated () , click it to show labels before you work with them. (This does not affect axis labels, which are always visible.)

Note, too, that standard and point labels can only be added to the [selected plot](#). Select the desired plot before using either of these tools.

Standard labels

Standard labels are similar to Post-it notes that you can pin on a plot. They are anchored to a spot on the plot where you click the tool and are scrolled with the plot. If the plot is zoomed, they remain the same size. If the plot is deleted, the label is also deleted.

To add a label to the [selected plot](#), make sure that labels are displayed (see above) and click the  toolbar button. This opens the **Label** dialog in which you can enter the text that you want to display. You can also specify the font and background color of the label, as well as its orientation and the alignment of the text within the label. When you click **OK** in the dialog, the mouse pointer indicates you note text, and clicking in the selected plot positions the label there.

To reposition an existing label, drag it around with the mouse.

To edit the text or style of a note, double-click it. This opens the **Label** dialog in which you can make changes as desired.

To delete a label, select it and click  in the toolbar.

Point labels

Point labels are very similar to standard labels, and most of the information given above also applies to them. The main difference is that point labels include a line which is anchored to a point of the plot. They also add a box around the label text.

Therefore you click twice when creating a point label: The first click positions the anchor point where the line starts, and the second click positions the box with the text.

When repositioning point labels, dragging the text repositions it but leaves the anchor point where it is, and dragging the point leaves the text where it is. Dragging the connecting line repositions all elements together.

Annotating axes

While labels cannot be attached to axes with the label tools, you can add titles or other descriptive text in the axis settings.

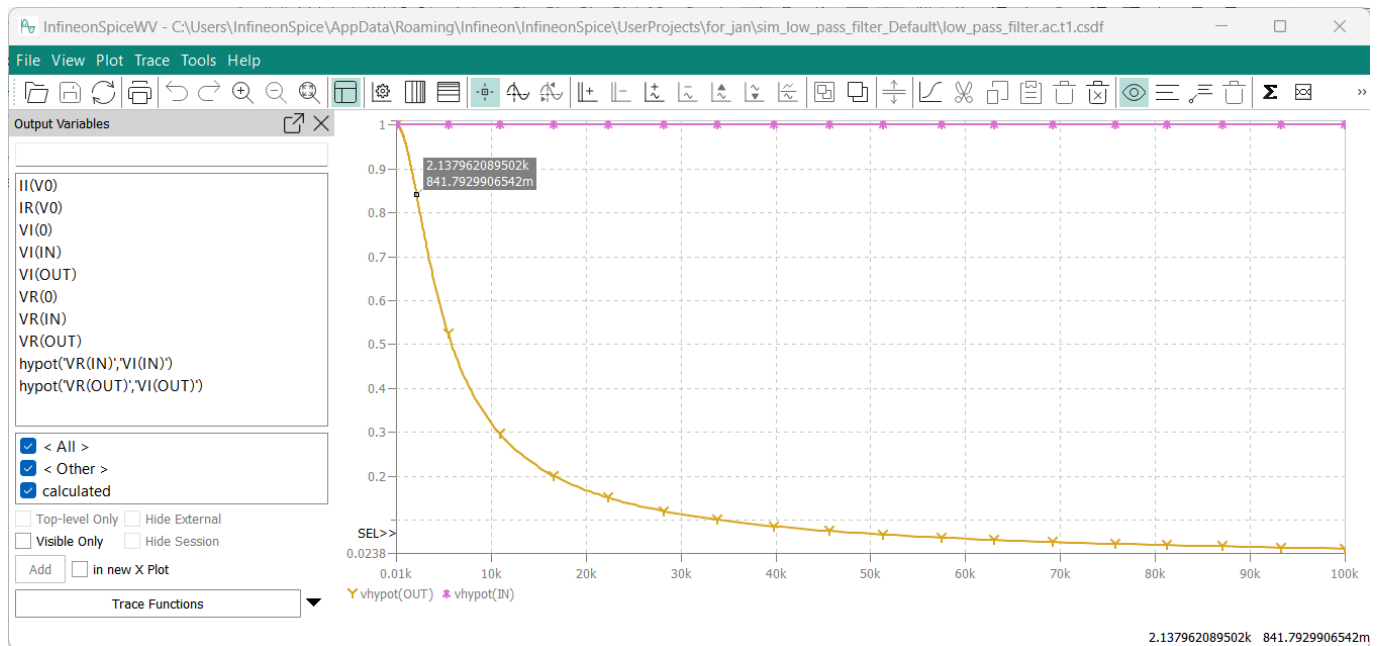
To do so, select the axis that you want to label and choose **Tools > Axis Settings** or click  in the toolbar. (For details on selecting an axis, see [Working with axes](#).) In the **Axis Settings** dialog that opens, check the **Custom** option in the **X Axis Label** or **Y Axis Label** section, respectively, and enter the text in the text field. If you want to show the custom text in place of (as opposed to in addition to) the standard axis label, uncheck the **Standard** option.

3.9.5 Evaluating AC analyses

AC analyses can be evaluated like other analyses but some special considerations apply.

Sample result of running an AC analysis, in this case using the low-pass filter schematic from the quick start project:

3 Using InfineonSpice



The plot shows the magnitudes of the IN and OUT signals.

Note that only the magnitudes for the IN and OUT signals were specified in the plot setup of the simulation. The real and imaginary values of the named voltages and currents of the schematic were automatically added in the **Output Variables** list.

Axes

When performing AC analyses with logarithmic range sweeps, you usually will want to switch the x-axes to logarithmic scaling by clicking .

Traces

Some aspects of the signals that you can display as separate traces often have vastly different amplitudes than others, for example, attenuation or phase. You may want to display such traces in separate plots by choosing **Plot > Scatter Traces in Plots**.

Adding signal aspects

For any named signal in the schematic, the real and imaginary values are automatically added to the simulation. Other aspects – magnitude, attenuation, or phase – can be specified explicitly. If you have not done so when setting up the simulation, they can also be added later.

To do so, choose **Trace > Freq Analysis** in the Waveform Viewer window. This opens the **Freq Analysis** dialog:

3 Using InfineonSpice

Click the **Select** button and choose the desired signal. In the **Function** section, click the aspect that you want to add. A name and alias are automatically entered; you can edit them if desired.

From the **Place** list, choose the plot to which to add the trace. You can also specify that a new plot is created for the trace.

Finally, click **OK** to create the trace. The trace appears in the specified plot.

3.10 Data sharing

InfineonSpice offers a variety of ways to import data, export data, and share data between projects and team members.

Importing components

InfineonSpice comes with a wide range of basic components and access to Infineon's online model store. Importing those components is described in [Managing Infineon and third-party components](#).

Importing projects

You can import projects that other users have exported using the **File > Import Project...** command. See [Importing a project](#) for more information.

Exporting projects

If you want to share projects with others, you can export them using the **Export** command from the context menu in the **Workspace** view. Exported projects are single files that can be imported by any InfineonSpice user. See [Exporting a project](#) for more information.

Sharing schematics and components

Schematics and components can be saved in custom containers, which can be shared among all your projects. See [Custom containers](#) for more information.

Creating graphics

You can create graphics from schematics in three ways:

- To create a PDF, choose **File > Print** and use the operating system's print-to-PDF driver.
- To create an SVG file, use **File > Export as SVG**.
- To copy the schematic as a graphic, use **Edit > Copy Schematic as Picture**.

Note that each of these methods creates graphics from the print area. Any elements you may have placed outside the print area are not included.

3.10.1 Custom containers

Custom containers are libraries of schematics and local components that you can share between projects.

While the components and schematics that make up a project are normally local to that project (except for the basic components built into InfineonSpice), you can create custom libraries of such elements that you can reuse between all of your projects.

Custom containers appear in the **Model Library** view's Custom Components section, from where you can add the items in them to your projects.

You can create custom containers, add items to them, and delete them:

- Creating a custom container
To create a custom container, right-click a schematic or local component in the **Workspace** view and choose **New Custom Container From**.

3 Using InfineonSpice

A dialog opens in which you can name the container. When you click **OK**, the container file is created in the project folder. The item from which you have created the container appears in the **Model Library** view's Custom Components section in the Custom (for local components) or TestBench (for schematics) section.

- Adding an item to a custom container.

To add a schematic or a local component from a project to an existing custom container, right-click it in the **Workspace** view and choose **Add to Existing Container**.

A dialog opens in which you can choose the container to which to add the item. When you click **OK**, the item is added to the container and appears in the appropriate subcategory of the **Model Library** view's Custom Components section.

- Removing an item from a custom container.

Once a schematic or local component was added to a container, it is not possible to remove it. As a workaround, you can create a new container with the same items as the current container, except the one that you want to remove. When you have added all desired items to the new container, you can delete the original container.

- Deleting a custom container.

To delete a custom container, right-click it in the **Model Library** view and choose **Remove** from the context menu.

3.10.2 Exporting a project

A project is exported using the Export command, making it available to other users.

Export a project when you want to make it available outside of your copy of InfineonSpice, for example, to give it to another user or to archive it as a backup

Steps

1. In the **Workspace** view, right-click the project that you want to export and choose **Export**. The **Export Project** dialog opens.
2. In the **Export location** field, specify where the exported project will be saved.
You can enter a path or choose the location by clicking **Browse**.
3. In the **Project Items** list, check the items you want to export and uncheck those that you do not want to include in the export.
If an item depends on other items, you can expand the item to show those other items. They can be checked or unchecked just like the main items.
It is possible that a main item is unchecked, but the dependency is checked and so will be included in the export.
4. If you want to exclude some profiles and simulation results from the export, select the item to which they belong in the **Project Items** list and uncheck the unwanted entries in the **Profiles and Results** list.
5. If you do not want to include any external files on which items in the project are dependent, uncheck them in the **List of files from external dependencies that will be included** list.
6. Click **OK**.

Results

The specified elements are exported as a project file (a Zip file with the .ifzip extension). These files can be imported by other users of InfineonSpice.

3.10.3 Importing a project

Projects are imported using the **Import Project** command or by drag and drop.

You can import projects that have been exported from InfineonSpice, for example, by other users to make them available to you.

Instead of using the menu command, as described below, you can also import a project by dragging it from the File Explorer to the **Workspace** view. This also opens the **Import Project** dialog, but you skip directly to step 5 of the procedure below.

Steps

- 1.** Choose **File > Import Project** .
The **Import Project** dialog opens.
- 2.** In the **File to import** section, specify the exported project that you want to import.
You can enter the path or click ... to choose a file. Only exported InfineonSpice projects (.ifzip) can be imported.
- 3.** In **New project name**, enter the name that the imported project will have in your workspace.
- 4.** In **Import Location**, choose where the project folder will be created.
If you choose **User space**, the folder will be create in the default location in your user folder where InfineonSpice also creates new projects.
If you choose **Custom location**, you can create the project folder in any location on your computer. In that case, you must specify the location either by entering the path in the text field or by clicking ... to choose a location.
If you specify a location on a file server or removable volume, you will not be able to access it if the server or volume is not connected.
- 5.** Click **OK**.

Results

The project is imported and appears as a folder in the **Workspace** view. You can work with it like with any other project.

4 InfineonSpice reference

This Reference guide gives definitions and explanations of terms that are in the entire InfineonSpice user guide.

4.1 Naming directories and files

Take note of the use of characters and rules at all times when naming directories and files

Directories

You can change directory names, but do not use spaces in the installation directory name. Also, do not use common illegal characters, such as: / : * ? " < > |

Files

The name may contain letters (a-z), numbers (0-9) and some special characters, such as:

& () + - = @ [] ^ ' { } ~. A dot (.) can also be used except at the end; an underscore (_) can be used except at the beginning of the name.

Refer to

- [Custom Installation](#)
- [Creating a new project](#)
- [Setting up a project](#)

4.2 Menus

A menu is a set of options presented to the user of a computer application to help them find information or execute a function. Menus are common in GUIs. Depending on what kind of content is displayed in the frontmost tab, different commands are available in the menu.

Open InfineonSpice and the following menus appear on the menu bar:

- [File menu](#)
- [View menu](#)
- [Options menu](#)
- [Window menu](#)
- [Help menu](#)

Additional menus can become visible if the frontmost tab is an editing tab of the appropriate type:

- [Edit menu](#)
- [Place menu \(schematics tab\)](#)

4.2.1 File menu

The **File** menu provides the usual options for managing documents and closing the application.

The **File** menu is a dynamic menu, that is, only commands that apply in the current situation are shown in the menu. See the descriptions of the individual commands for information on when they are available.

New

This submenu contains commands for creating new documents of various types:

New Project

Choosing this command opens the **New Project** dialog, in which you can specify the name of the new project.

Clicking **OK** in that dialog creates the new project, which appears as a folder in the **Workspace** view.

New Schematic

Choosing this command opens the **New Schematic** dialog, in which you can specify the name of the new schematic.

4 InfineonSpice reference

Clicking **OK** in that dialog creates the new schematic, which appears as a document in the current project in the **Workspace** view and is opened in the work area.

This command can only be chosen when a project is selected in the **Workspace** view.

New Library

Choosing this command opens the **New Library** dialog, in which you can specify the name of the new library.

Clicking **OK** in that dialog creates the new library, which appears as a document in the current project in the **Workspace** view and is opened in the work area.

This command can only be chosen when a project is selected in the **Workspace** view.

New Symbol

Choosing this command opens the **New Symbol** dialog, in which you can specify the name of the new symbol.

Clicking **OK** in that dialog creates the new symbol, which appears as a document in the current project in the **Workspace** view and is opened in the work area.

This command can only be chosen when a project is selected in the **Workspace** view.

New Local Component

Choosing this command opens the **New Device** dialog, in which you can specify the name of the new component.

Clicking **OK** in that dialog creates the new component, which appears as a document in the current project in the **Workspace** view and is opened in the work area.

This command can only be chosen when a project is selected in the **Workspace** view.

Import Project

Choosing this command lets you import projects that do not yet appear in your InfineonSpice workspace.

Choosing the command opens the **Import Project** dialog, in which you can import previously exported InfineonSpice projects. For details, see [Import Project dialog](#).

The imported project appears as a folder in the **Workspace** view.

Check and Save

Choosing this command verifies the formal correctness of the active schematic and saves it.

If any issues are found during the verification, they are written to the **Output** view.

This command is only active when the active tab in the work area contains a schematic.

Save

Choosing this command saves the changes that have been made in the active view, for example, **Simulation Environment**.

This command is only available if the active view contains changes that can be saved.

Save As

The **Save As** command works the same as [Save](#) but lets you specify a new name for the saved document.

Print Preview

Choosing this command opens the [Print Preview](#) window in which you can specify various aspects of how the currently active schematic is printed.

This command is available only if the active tab is a schematic editor.

Print

Choosing this command opens the operating system's **Print** dialog. For details on this dialog, see the operating system documentation.

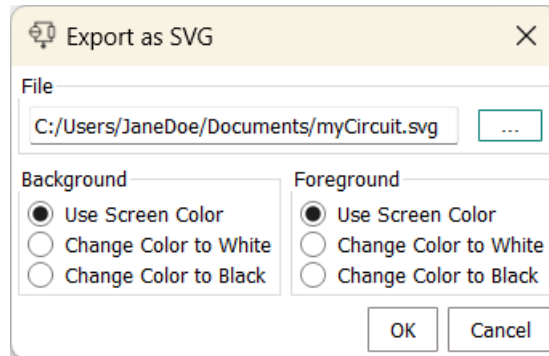
Clicking **Print** in the dialog prints the schematic according to the settings made in the [Print Preview](#) window.

This command is available only if the active tab is a schematic editor.

Export as SVG

4 InfineonSpice reference

Choosing this command opens the **Export as SVG** dialog.



The dialog has three areas:

File

Enter here the path and name under which you want to save the SVG version of the schematic.

Clicking the ... button opens an operating system dialog in which you can choose a directory and specify the name.

Background

You can choose the background color for the exported SVG graphic. You can specify that the same color as on screen, white, or black are to be used.

Foreground

You can choose the color for the schematic elements in the exported SVG graphic. You can specify that the same colors as on screen, white, or black are to be used.

This command is available only if the active tab is a schematic editor.

Save All

Choosing this command saves all changes in open views as if the [Save](#) command had been used for each of them.

Exit

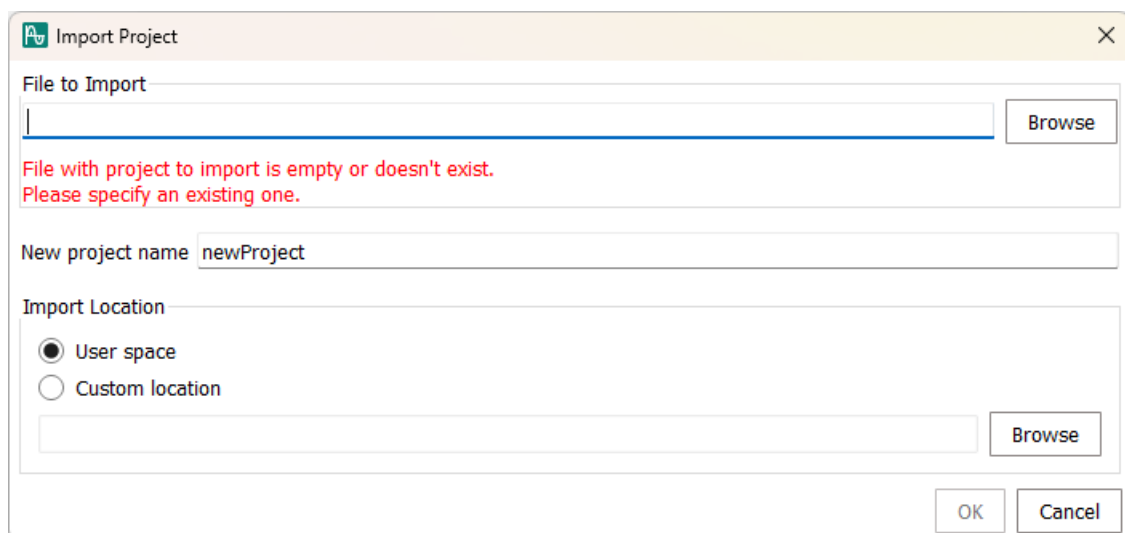
Closes InfineonSpice.

If any documents contain unsaved changes, you are prompted to save them.

4.2.1.1 Import Project dialog

The **Import Project** dialog lets you import existing projects into your workspace.

The dialog is opened by choosing **File > Import Project**.



4 InfineonSpice reference

File to Import

The project file that you want to import. You can import any exported InfineonSpice project file (.ifzip file).

You can specify the file to import by entering its full path into the text field. Alternatively, you can click the **Browse** button to the right, which opens Windows' file dialog, and choosing the desired file.

New project name

The name that the imported project has in your workspace.

Enter the desired name without any filename extensions.

Import location

The place where you want to store the imported project:

User space

The project is stored in InfineonSpice's application data folder, in the same location where projects you create in the app are stored.

Custom location

The project is stored in a place you specify.

You can specify the desired directory by entering its full path into the text field. Alternatively, you can click the **Browse** button to the right, which opens Windows' file dialog, and choose the desired folder.

The project folder is created inside the specified directory.

OK

Imports the specified project and closes the dialog.

The project appears in the **Workspace** view. The project folder on disk is created in the location you have specified in **Import location**.

Cancel

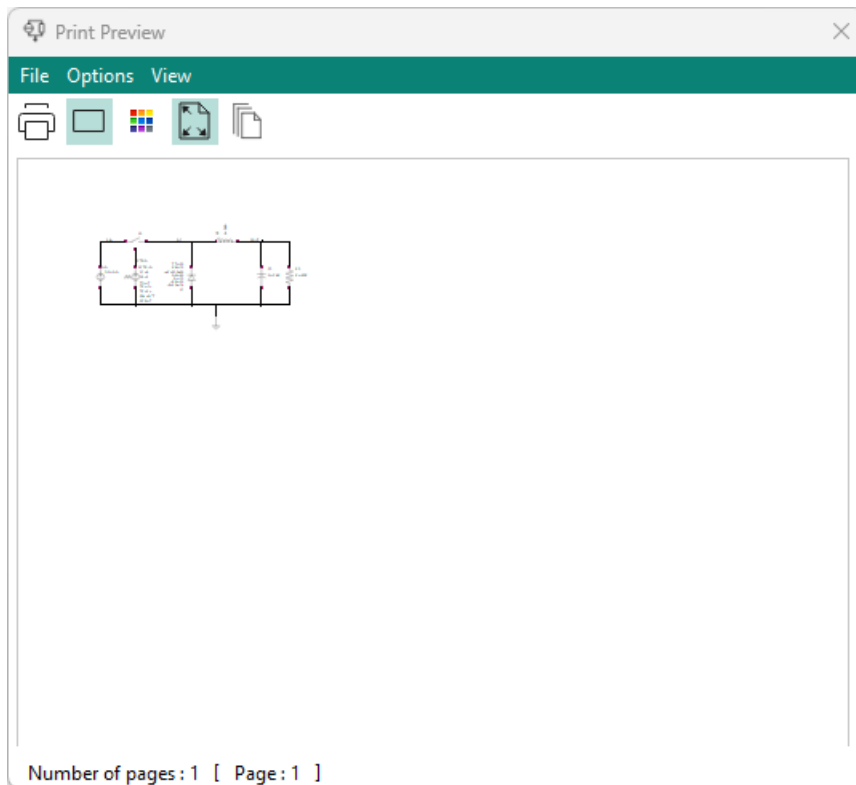
Closes the dialog without importing anything.

4.2.1.2 Print Preview window

The **Print Preview** window, shows how the printout of a schematic will look and lets you adjust output parameters.

The window is opened by choosing **File > Print Preview** in a schematic-editing view.

4 InfineonSpice reference



Menus

The three menus in the **Print Preview** window let you access the same commands as the toolbar, in addition to the **Close** command.

File

Print

Opens the Windows print dialog, where you can print the schematic with the current settings from this window.

For details about the dialog, see the documentation for Windows and for your printer.

Close

Closes the **Print Preview** window without printing anything.

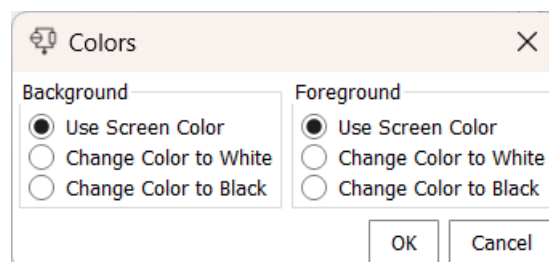
Options

Frame

Lets you toggle a frame that is printed around the schematic.

Colors

Opens the **Colors** dialog in which you can specify in which colors to print the foreground (that is, the elements of the schematic) and the background.



4 InfineonSpice reference

If you choose **Use Screen Color**, elements are printed in the same colors in which they are displayed in the schematic editing tab.

View

Fit to Page

Toggles the resizing of the schematic printout between the schematic's original size and a reduced size that fits on one printer page.

If the schematic in its original size is smaller than a printer page, it is not resized.

All Pages

Toggles the dialog between displaying one page and all pages of the schematic printout.

Toolbar

The toolbar of the **Print Preview** window lets you access the same commands as the menus, except for the **Close** command.



Opens the Windows print dialog, where you can print the schematic with the current settings from this window.

For details about the dialog, see the documentation for Windows and for your printer.



Lets you toggle a frame that is printed around the schematic.



Opens the **Colors** dialog which is described in the [Colors](#) command, above.



Toggles the resizing of the schematic printout between the schematic's original size and a reduced size that fits on one printer page.

If the schematic in its original size is smaller than a printer page, it is not resized.



Toggles the dialog between displaying one page and all pages of the schematic printout.

Status bar

The status bar at the bottom of the window displays the number of printed pages the schematic will require using the current settings, and it displays the number of the currently shown page.

4.2.2 Edit menu

The **Edit** menu contains the usual function for reverting actions, controlling the clipboard, and for selecting, finding, and deleting items. In some views, the menu contains additional commands for manipulating the views' contents.

The **Edit** menu is a dynamic menu that contains different commands depending on the frontmost editing tab. Furthermore, the menu is hidden completely when no editing tab is open.

Move

Lets you move elements in a schematic.

Choosing this command turns the cursor into a cross. Click-dragging this cursor in a schematic selects all elements that are completely surrounded by the dragging rectangle. After releasing the mouse, you can:

4 InfineonSpice reference

- Click-drag again to move the selected elements to another position in the schematic.
- Press the Escape key to fix the element's position at the current location.

This command is available only in schematic-editing tabs.

Cut

Copies the selected element or text to the clipboard and removes it from the editor.

Copy

Copies the selected element or text to the clipboard.

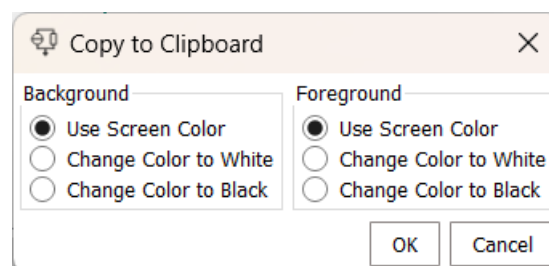
In a schematic-editing tab, choosing this command turns the cursor into a cross. Click-dragging this cursor in a schematic selects all elements that are completely surrounded by the dragging rectangle. After releasing the mouse, the selected elements are copied to the clipboard. You can terminate the copying process by pressing the Escape key.

In a symbol-editing tab, the currently selected element is copied. In a text view or tab, the selected text is copied.

Copy Schematic as Picture

Copies the entire schematic to the clipboard as a bitmap graphic.

Choosing the command opens the **Copy to Clipboard** dialog, in which you can modify the colors of the copied bitmap:



If you choose **Use Screen Color**, the elements or the background, respectively, are copied in the colors with which they are displayed on screen.

This command is available only in schematic-editing tabs.

Paste

Inserts the current content of the clipboard into the frontmost editing tab.

In text tabs, the text appears at the text insertion mark. In graphics tabs, the cursor takes on the shape of the pasted element. Clicking the mouse at the desired location in the editing tab places the element. You can click multiple times to place additional copies. To terminate the pasting mode, press the Escape key.

Delete

Removes the selected element or text from the document without placing it on the clipboard.

In a schematic-editing tab, choosing this command turns the cursor into a cross. Click-dragging this cursor in a schematic selects all elements that are completely surrounded by the dragging rectangle. After releasing the mouse, the selected elements are deleted. You can terminate the copying process by pressing the Escape key.

In a symbol-editing tab, the currently selected element is deleted. In a text-editing view or tab, the selected text is deleted.

Undo

Reverts the most recent change.

You can undo multiple actions by choosing **Undo** repeatedly. Not all actions can be undone.

Redo

Reapplies the most recently undone action.

You can redo multiple actions if you undid them before.

4 InfineonSpice reference

Redo is available only when the latest action was **Undo**. If you perform other editing actions after choosing **Undo**, you cannot redo the undone action.

Select All

Select the entire content of the frontmost editing tab.

Page Setup

Specifies the page size and frame dimensions of the schematic.

Choosing the commands opens the [Schematic Page Setup dialog](#).

This command is available only in schematic-editing tabs.

Properties

Displays the properties of the selected components.

Depending on the tab and the selected element, different dialogs open:

- In a schematics tab: [Properties dialog](#)
- In a symbol tab with selected pin or shape: [Attributes dialog](#)
- In a symbol tab with selected substitution field: [Substitution Field dialog](#)
- In a symbol tab with selected drawing text: [Text Attributes dialog](#)

This command is not available in text-editing views or tabs.

Select Entire Net

Selects all wires that are connected to the currently selected wire.

This command is available only if a wire in a schematic is selected. Choosing it selects all wires that are connected to the selected wire without crossing a component, in addition to the currently selected wire.

Send Backward

Puts the selected element behind all other elements in the graphic.

This command is available only in symbol-editing tabs.

Bring Forward

Puts the selected element in front of all other elements in the graphic.

This command is available only in symbol-editing tabs.

Rotate

Rotates the selected element 90 degrees counterclockwise.

This command is not available in text-editing views or tabs.

Mirror (Vert.)

Mirrors the selected element vertically (flipping top and bottom).

This command is not available in text-editing views or tabs.

Mirror (Horiz.)

Mirrors the selected element horizontally (flipping left and right).

This command is not available in text-editing views or tabs.

Net Color

Lets you modify the colors of wires.

This submenu is available only in schematic-editing tabs.

Highlight Net

Colors the selected wire and all connected wires in the highlight color, that is, the color that indicates a selected wire.

Note that the wires retains this color even if they are no longer selected. This may lead to confusion about which wires currently are selected.

This command is available only in schematic-editing tabs.

Set Net Color

4 InfineonSpice reference

Lets you assign any desired color to the selected wire and all connected wires.

Choosing the command when one or more wires within the same net are selected opens [a dialog](#).

If wires from multiple nets are selected, the [Net Colors](#) dialog opens instead.

This command is available only in schematic-editing tabs.

Reset Net Color

Colors the selected wire and all connected wires in their default colors.

This command is available only in schematic-editing tabs.

Reset All Net Colors

Colors all wires in the schematic in their default colors.

This command is available only in schematic-editing tabs.

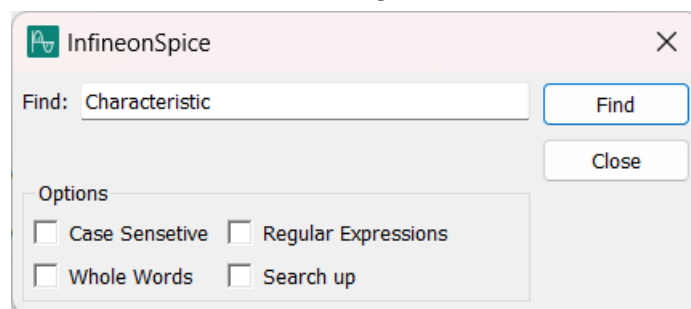
Update Pins from View

(Currently under review.)

Find

Lets you find text in the frontmost tab.

Choosing the command opens the **InfineonSpice** dialog:



The dialog contains:

Find:

The search text, that is, text you are looking for.

Find

Selects the next occurrence of the search text.

You can click **Find** repeatedly to cycle through all occurrences. Searches are wrap-around, that is, if the end of the text is reached, the search continues from the top.

Close

Closes the dialog.

Case-sensitive

If this option is checked, only text with the same uppercase and lowercase letters as the search text is found.

For example, searching with this option for “top” finds “stop” but not “Top-notch”.

Whole words

If this option is checked, only entire words that match the search text are found, not parts of words.

For example, searching with this option for “will” finds “will” or “Will” but not “swill” or “William”.

Regular expression

If this option is checked, the search text is interpreted as a regular expression and all text that matches the expression is found.

For example, searching with this option for “a.t” finds “ant” or “art” but not “at” or “absent”.

Search up

4 InfineonSpice reference

This option reverses the search direction.

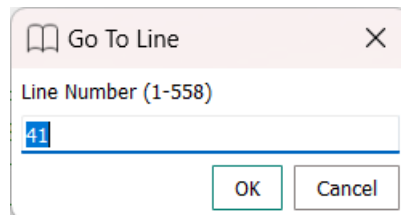
When it is checked, clicking **Find** selects the search match before the current one. Searches that reach the top of the text continue at the bottom.

This command is available only in text-editing tabs.

Go to Line

Lets you jump to a specific line in the text.

Choosing the command opens the **Go to Line** dialog:



Clicking **OK** in this dialog puts the text insertion mark at the beginning of the specified line.

This command is available only in text-editing tabs.

Replace

Lets you find and replace text in the frontmost tab.

This dialog is similar to the dialog described in [Find](#), above, but has three additional elements:

Replace with

The text which you want to insert into the document in place of found text.

Replace

Clicking this button removes the selected text and inserts the text from the **Replace with** field in its place.

Replace All

Clicking this button removes all instances of the search text and inserts the text from the **Replace with** field in their places.

This command is available only in text-editing tabs.

4.2.2.1 Schematic Page Setup dialog

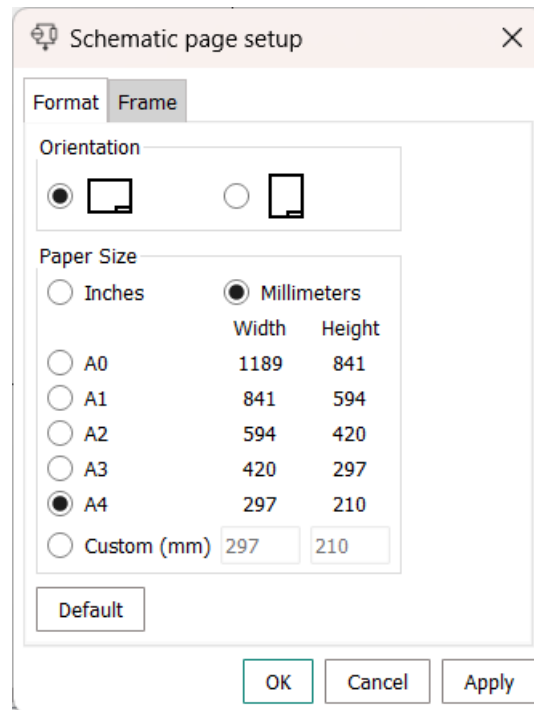
This dialog lets you specify the page size and frame location for schematics.

The dialog contains two tabs, **Format** for the page format and **Frame** for the frame position. The buttons at the bottom of the dialog apply to both frames.

The dialog is opened by choosing **Edit > Page Setup** in a schematic-editing view.

4 InfineonSpice reference

Format tab



Orientation

Choose between landscape (wide) and portrait (tall) orientation of the page. The choice is reflected in the page width and height displayed in the **Paper Size** section.

Paper Size

Inches

Clicking **Inches** displays US paper sizes and converts the displayed paper dimensions to inches.

Millimeters

Clicking **Millimeters** displays ISO standard paper sizes and converts the displayed paper dimensions to millimeters.

E, D, C, B, A

Choosing one of these options set the paper size to the chosen format. The page dimensions are displayed in the **Width** and **Height** columns.

These options are displayed when **Inches** is chosen.

A0, A1, A2, A3, A4

Choosing one of these options set the paper size to the chosen format. The page dimensions are displayed in the **Width** and **Height** columns.

These options are displayed when **Millimeters** is chosen.

Custom

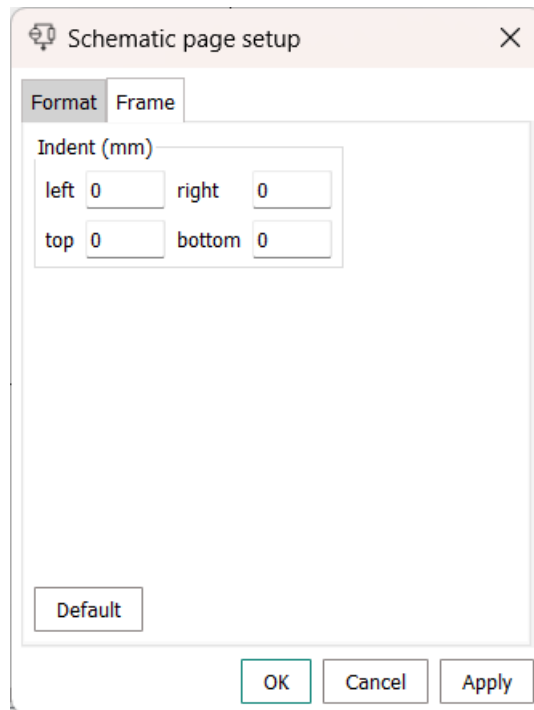
Choosing **Custom** lets you freely specify the page size by entering the desired dimensions into the two text fields in the **Width** and **Height** columns.

Default

Resets the page size settings to the standard values.

4 InfineonSpice reference

Frame tab



In this tab, you can specify the distance in millimeters between the page border and the frame around the schematic.

Left, Right, Top, Bottom

Specifies the distance between the respective side of the frame and the page border.

For example, entering 12 into the **Left** field causes the left side of the frame to be drawn 12 mm away from the page border.

Default

Resets the frame settings to the standard values.

Common buttons

OK

Closes the dialog and applies the specified page size and frame settings to the current schematic.

Cancel

Closes the dialog without changing the page size and frame settings of the schematic.

Apply

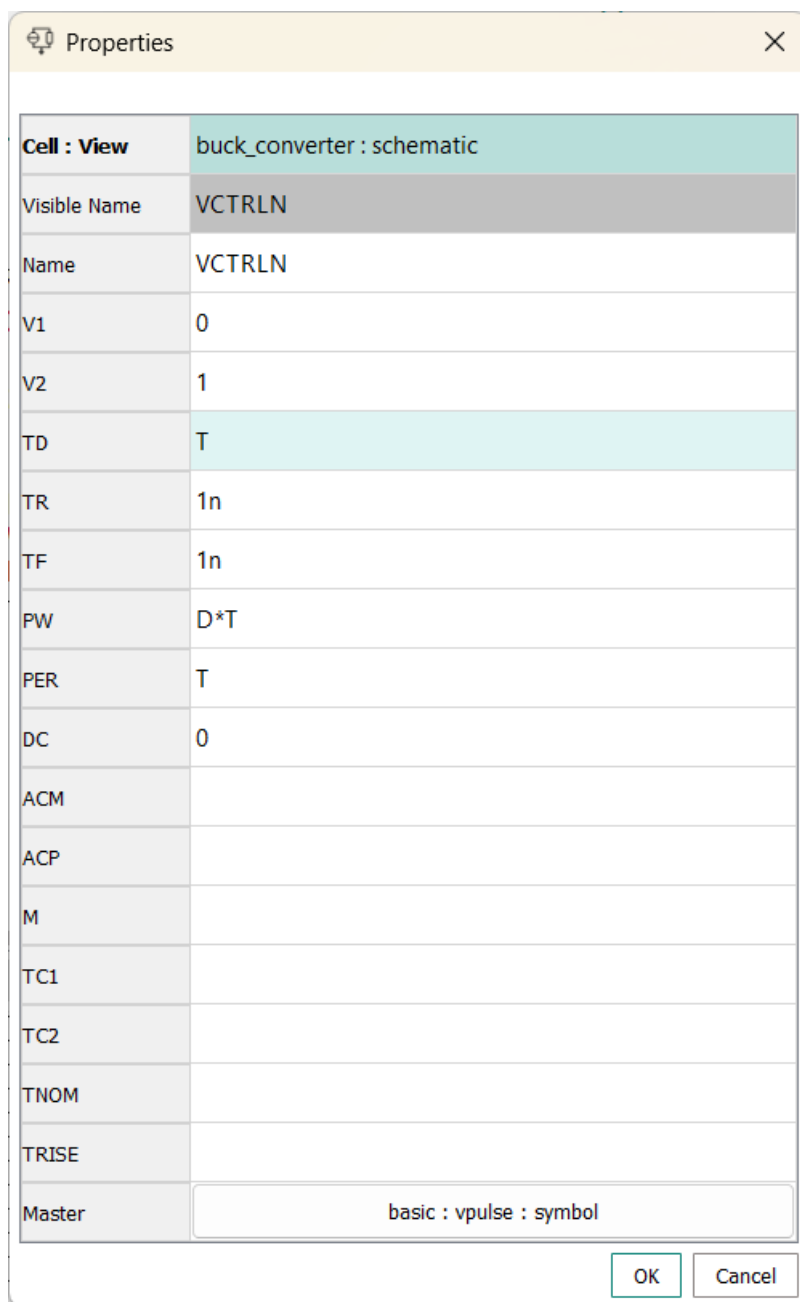
Applies the specified page size and frame settings to the current schematic and leaves the dialog open.

4.2.2.2 Properties dialog

The **Properties** dialog lets you view and modify properties of components in a schematic.

The dialog is opened by selecting the desired component in a schematic and choosing **Edit > Properties**.

4 InfineonSpice reference



Cell : View	buck_converter : schematic
Visible Name	VCTRLN
Name	VCTRLN
V1	0
V2	1
TD	T
TR	1n
TF	1n
PW	D*T
PER	T
DC	0
ACM	
ACP	
M	
TC1	
TC2	
TNOM	
TRISE	
Master	basic : vpulse : symbol

OK Cancel

The dialog contains a column with cell names along the left side and one column showing the associated values.

You can select cells by clicking them. You can edit white (but not light or dark gray) cells by double-clicking them.

Buttons

OK

Applies the changes made in the dialog to the selected components and closes the dialog.

Cancel

Asks you whether you want to save any changes you have made before closing the dialog.

4.2.2.3 Attributes dialog

The **Attributes** dialog lets you view and modify properties of elements in a symbol.

The dialog is opened by selecting the desired element in a symbol view and choosing **Edit > Properties**. The command is not available when multiple elements are selected.

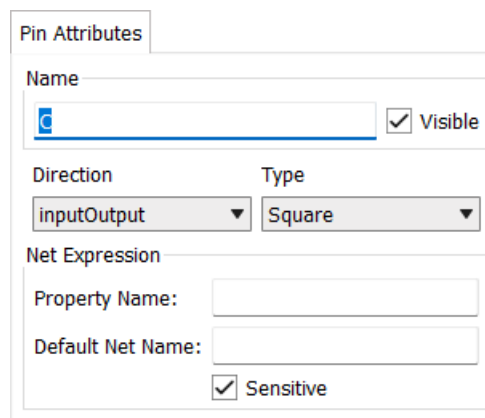
Depending on the type of the selected component, different versions of the dialog are shown:

- Pin: Attributes dialog with [Pin Attributes tab](#)
- Substitution field: [Substitution Field dialog](#)
- Drawing text: [Text Attributes dialog](#)
- Closed shapes: **Attributes** dialog with [Pen Attributes](#) and [Brush Attributes](#) tabs
- Line shapes: **Attributes** dialog with [Pen Attributes](#) tab
- Arrows: **Attributes** dialog with [Arrow Attributes](#), [Pen Attributes](#), and [Brush Attributes](#) tabs

Clicking the **OK** button applies the settings in the dialog to the selected element and closes the dialog. Clicking **Cancel** closes the dialog without making changes to the element.

Pin Attributes tab

The **Pin Attributes** tab lets you modify the properties of a pin.



Name

The name of the pin.

Visible

Whether the pin name is visible in the symbol.

Direction

The direction of the pin: input output, or both.

Type

Whether the pin is represented in the symbol by a square or a circle.

Property Name

(Currently under review.)

Default Net Name

The default name for the wires that are connected to the pin.

This information is optional.

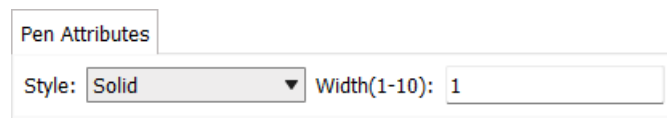
Sensitive

(Currently under review.)

Pen Attributes tab

The **Pen Attributes** tab lets you modify the stroke properties of a closed shape, a line, or an arrow.

4 InfineonSpice reference



Pen Attributes

Style: **Solid** Width(1-10): **1**

Style

Whether the shape has an uninterrupted (solid) stroke or the stroke is composed of dots, dashes, or a combination of the two.

Width

The width of the stroke.

Brush Attributes tab

The **Brush Attributes** tab lets you modify the fill properties of a closed shape or an arrow.



Brush Attributes

Brush Style: **Hollow**

Brush Style

Whether the selected shape is just an outline (hollow) or filled (solid).

Arrow Attributes tab

The **Arrow Attributes** tab lets you modify the dimensions of an arrow tip.



Arrow Attributes

Length: **10** Width: **5**

Length

The length of the tip of the arrow. Higher numbers create longer tips.

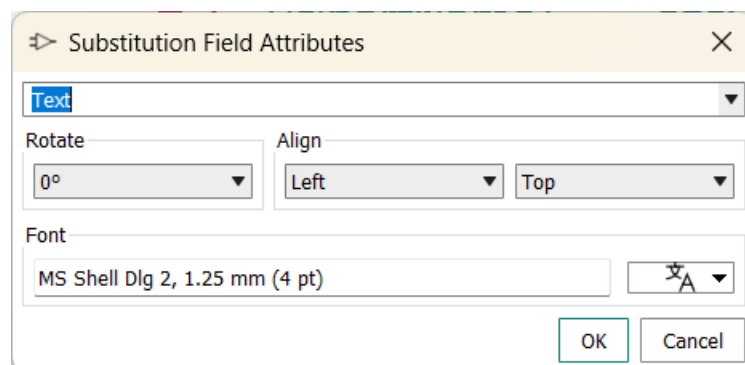
Width

The width of the tip of the arrow. Higher numbers create tips with wider bases.

4.2.2.4 Substitution Field dialog

The **Substitution Field** dialog lets you view and modify properties of substitution fields in a symbol.

The dialog is opened by selecting a substitution field in a symbol view and choosing **Edit > Properties**. The command is not available when multiple fields are selected.



Substitution Field Attributes

Text

Rotate: **0°** Align: **Left** **Top**

Font: **MS Shell Dlg 2, 1.25 mm (4 pt)**

OK Cancel

Text field

The text field at the top of the dialog contains the name of the substitution field.

4 InfineonSpice reference

The menu at the right of the field lets you choose the type of text that is being specified:

Logical label

(Currently under review.)

Instance label

(Currently under review.)

Analog device annotate

(Currently under review.)

Rotate

The orientation of the substitution field in the symbol.

Align

The alignment along the left-right and top-bottom axes of the field text in relation to its point of origin.

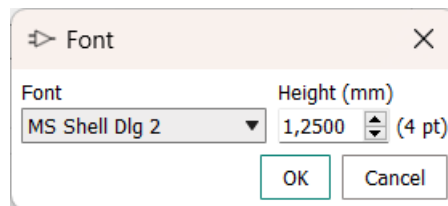
Font

The font in which the field text is shown in the symbol.

The menu at the right of the dialog section lets you modify the font:

Choose Font

Opens the **Font** dialog:



The **Font** menu lets you select the desired font from the fonts installed on your computer. The **Height** field lets you set the desired font size.

Clicking **OK** applies the current setting to the **Substitution Field Attributes** dialog and closes the **Font** dialog. Clicking **Cancel** closes the **Font** dialog without making any changes in the **Substitution Field Attributes** dialog.

Use Default Font

Sets the font information to the default font.

OK

Applies the settings made in the dialog to the substitution field and closes the dialog.

Cancel

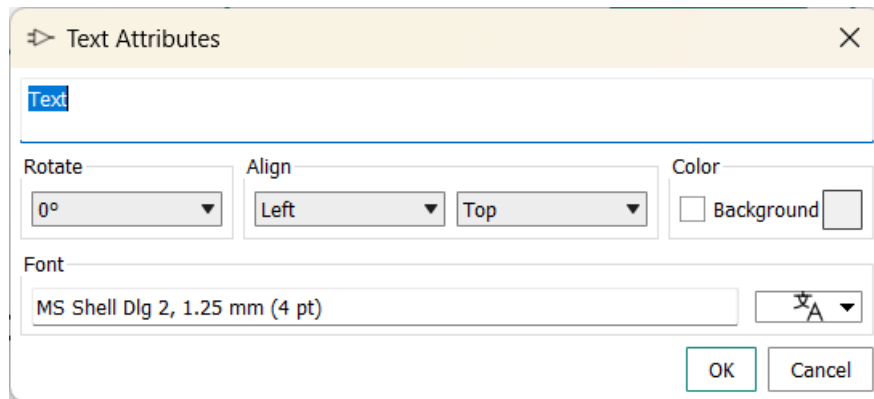
Closes the dialog without making any changes to the substitution field.

4.2.2.5 Text Attributes dialog

The **Text Attributes** dialog lets you view and modify properties of display text in a symbol.

The dialog is opened by selecting the desired text element in a symbol view and choosing **Edit > Properties**. The command is not available when multiple text elements are selected.

4 InfineonSpice reference



Text field

The text field at the top of the dialog contains the text that is being displayed.

Rotate

The orientation of the text in the symbol.

Align

The alignment along the left-right and top-bottom axes of the text in relation to its point of origin.

Color

Whether the text has a background.

If **Background** is checked, the text is displayed on a gray background. Otherwise, a transparent background is used.

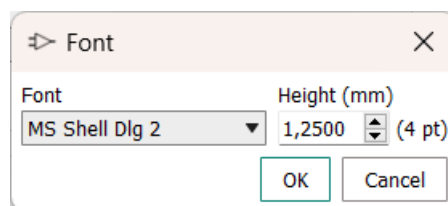
Font

The font in which the text is shown in the symbol.

The menu at the right of the dialog section lets you modify the font:

Choose Font

Opens the **Font** dialog:



The **Font** menu lets you select the desired font from the fonts installed on your computer. The **Height** field lets you set the desired font size.

Clicking **OK** applies the current setting to the **Text Attributes** dialog and closes the **Font** dialog.

Clicking **Cancel** closes the **Font** dialog without making any changes in the **Text Attributes** dialog.

Use Default Font

Sets the font information to the default font.

OK

Applies the settings made in the dialog to the display text and closes the dialog.

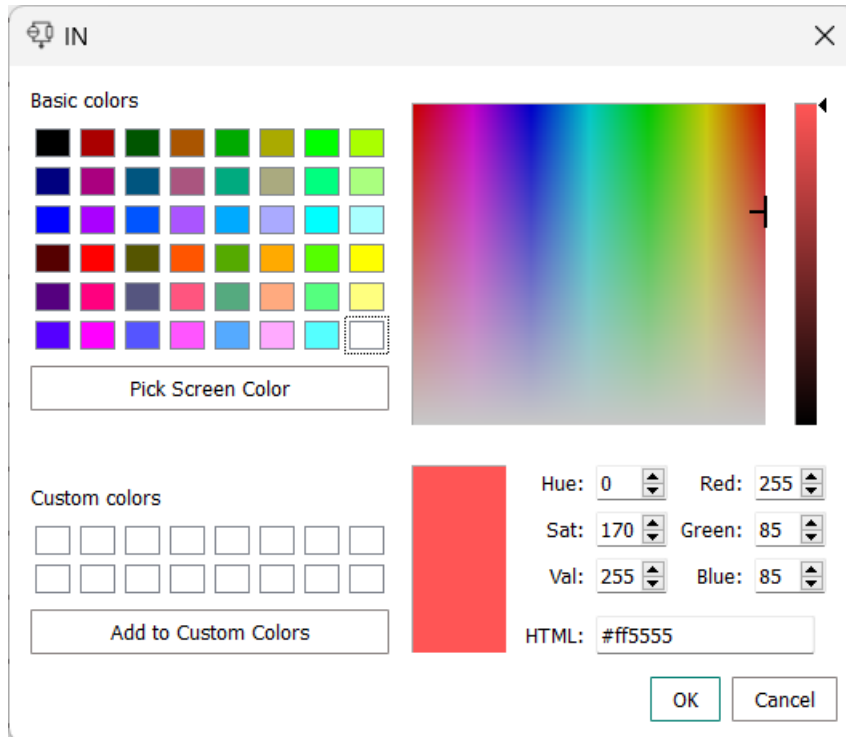
Cancel

Closes the dialog without making any changes to the display text.

4.2.2.6 Net color dialog

The net color dialog lets you specify a color for a selected wire or net.

The dialog is opened by selecting one or more connected wires and choosing **Edit > NetColor > Set Net Color**. (If you select multiple unconnected wires, the [Net Colors](#) dialog opens instead.)



This dialog is, in principle, a standard color picker.

Basic colors

This is a grid of predefined colors. Clicking any color assigns it to the color swatch (see below).

Pick Screen Color

Clicking this button turns the mouse pointer into a cross-hairs. Clicking the mouse anywhere on screen turns the color swatch (see below) into the color at that location.

Custom colors

This grid can be filled with colors you specify yourself. Any color saved as a custom color is thereafter available for assigning to wires in the same way as basic colors.

Color fields

Two color fields are located at the top left of the dialog. Clicking anywhere within these fields turns the color swatch (see below) into the color at that location.

Color swatch

The large rectangular color swatch at the bottom center of the dialog indicates the color that will be assigned to the selected wires by clicking **OK**.

Numerical controls

The numerical controls at the bottom right of the dialog let you specify the color of the color swatch by its color space coordinates. You can choose between:

- Hue, saturation brightness: Use the **Hue**, **Sat**, and **Val** fields.
- RGB: Use the **Red**, **Green**, and **Blue** fields.
- Web colors: Enter the six-digit hexadecimal number for the color (prefixed with #) into the **HTML** field.

Changing the color by any of these methods dynamically adapts the values for the other methods.

OK

4 InfineonSpice reference

Clicking **OK** assigns the color displayed in the color swatch to the selected wires and closes the dialog.

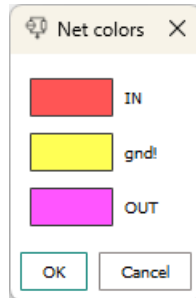
Cancel

Clicking **Cancel** closes the dialog without changing the color of the wires.

4.2.2.7 Net Colors dialog

The **Net Colors** dialog shows the colors of all selected wires and lets you modify them.

The dialog is opened by selecting wires from multiple nets and choosing **Edit > NetColor > Set Net Color**. (If you select only one wire or wires from just one net, the **net color** dialog opens instead.)



The dialog displays all selected nets and the colors that have been assigned to them.

Clicking any of the swatches opens the **net color** dialog in which you can modify the color.

Clicking **OK** assigns the colors that are shown in the dialog to the respective nets.

Clicking **Cancel** closes the dialog without recoloring the nets.

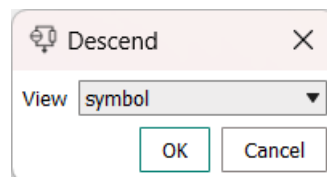
4.2.3 View menu

The commands in the **View** menu let you configure which views and toolbars are visible onscreen. You can also enlarge and reduce tab contents.

Descend Hierarchy

Lets you open a component in a new symbol or library tab.

Selecting a component and choosing **Descend Hierarchy** opens the **Descend** dialog.



The dialog contains a menu from which you can choose whether you want to view the symbol or the library of the component. Note that no library tab is available for some basic components.

Clicking **OK** opens the symbol tab or library tab (depending on your choice from the menu) and closes the dialog.

Clicking **Cancel** closes the dialog without opening a new view.

This command is only available in schematic tabs.

Zoom In

Displays the content of the active editing tab in a larger size.

This command is not available in text views and tabs.

Zoom Out

Displays the content of the active editing tab in a larger size.

This command is not available in text views and tabs.

Zoom Region

Lets you focus the active editing tab on a chosen part of the graphic.

4 InfineonSpice reference

Choosing **Zoom Region** turns the mouse pointer into a cross-hairs. Selecting a part of the graphic with the mouse zooms in on the part so that it fills the view.

This command is available only in schematics tabs.

Zoom All

Zooms out in the active editing tab so that the entire graphic is visible.

This command is available only in schematics tabs.

Reset View Layout

Returns all views to their default sizes and positions.

Workspace

Toggles the visibility of the **Workspace** view.

Model Libraries

Toggles the visibility of the **Model Libraries** view.

Simulation Environment

Toggles the visibility of the **Simulation Environment** view.

By default, this view is shown in the same frame as the **Properties** view as a separate tab.

Output

Toggles the visibility of the **Output** view.

Properties

Toggles the visibility of the **Properties** view.

By default, this view is shown in the same frame as the **Simulation Environment** view as a separate tab.

Workspace Toolbar

Toggles the toolbar of the **Workspace** view on and off.

Model Libraries Toolbar

Toggles the toolbar of the **Model Libraries** view on and off.

Schematic Toolbar

Toggles the toolbar of schematics tabs on and off.

Choosing this command affects all open schematics tabs.

Symbol Toolbar

Toggles the toolbar of symbol tabs on and off.

Choosing this command affects all open symbol tabs.

Status Bar

Toggles the status bar in the lower right corner of the InfineonSpice window on and off.

4.2.4 Place menu (schematics tab)

The **Place** menu of a schematics tab lets you put components and other elements into a schematic.

(For information on the **Place** menu in symbol tabs, see [Place menu \(symbol tab\)](#).)

Choosing a command from the menu turns the mouse pointer into a representation of the corresponding component. Clicking the mouse in the schematic places the component in that location. You can continue clicking to place additional components of the selected type. To stop placing this type of component, choose another command from the **Place** menu or press the Escape key.

The **Wire Name** and **Text** commands work in the same way but first open the **Wire Name** and **Place Text** dialog, respectively.

The **Line** and **Rectangle** commands work similar to component-placing commands but require two clicks (for the start end the end of the shape) for each placed element.

Select

4 InfineonSpice reference

Switches to the select tool, which lets you select components in the schematic.

Ground

Lets you place grounds in the schematic.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Resistor (EU)

Lets you place resistors in the schematic using a symbol that is common in the EU.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Resistor (US)

Lets you place grounds in the schematic using a symbol that is common in the US.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Capacitor

Lets you place capacitors in the schematic.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Inductor

Lets you place inductors in the schematic.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Current Source (DC)

Lets you place DC current sources in the schematic.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Voltage Source (DC)

Lets you place DC voltage sources in the schematic.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Diode

Lets you place diodes in the schematic.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Wire (Narrow)

Lets you place narrow wires in the schematic.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Wire (Wide)

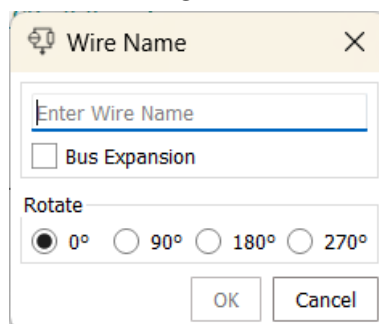
Lets you place wide wires in the schematic.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Wire Name

Lets you name wires in the schematic.

Choosing the command opens the **Wire Name** dialog:



Enter Wire Name

The name that you want to give the wire.

4 InfineonSpice reference

Bus Expansion

Rotate

The orientation of the text in the schematic graphic.

OK

Clicking **OK** closes the dialog and turns the mouse pointer into a representation of the wire name. Clicking a wire applies the name to it.

Choose a different command from the **Place** menu or press the Escape key to stop naming wires.

Cancel

Clicking **Cancel** closes the dialog and chooses the select tool (as if you had chosen **Select** from the **Place** menu).

Solder Dot

Lets you place solder dots in the schematic.

Choose a different command from the **Place** menu or press the Escape key to stop placing solder dots.

Line

Lets you place lines in the schematic. Click once for placing the start of the line, and click again to place the end.

Choose a different command from the **Place** menu or press the Escape key to stop placing lines.

Rectangle

Lets you place rectangles in the schematic. Click once for placing one corner of the rectangle, and click again to place the diagonally opposite corner.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Text

Lets you place text in the schematic.

Choosing the command opens the [Place Text dialog](#).

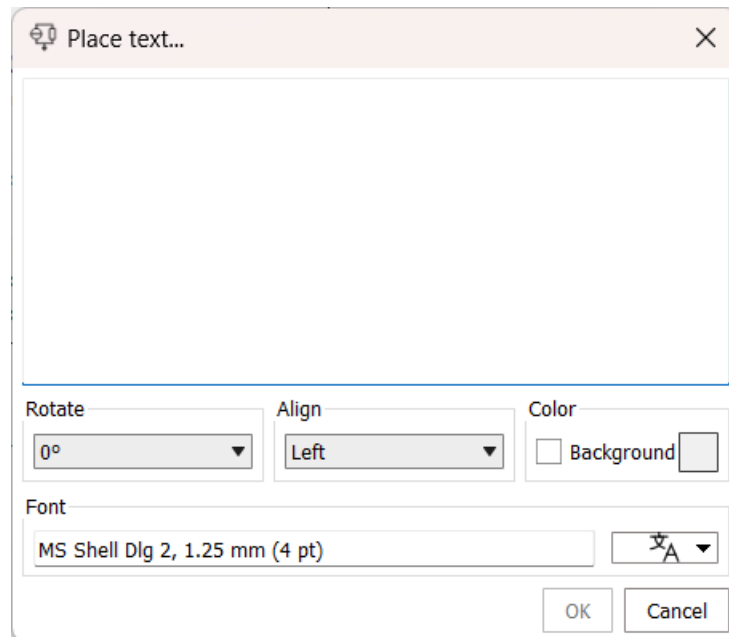
Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

4.2.4.1 Place Text dialog

The **Place Text** dialog lets you enter, modify, and format text that you want to place in a schematic.

The **Place Text** dialog is opened by choosing **Place > Text** or by clicking the Drawing Text tool () in the toolbar of a schematic tab.

4 InfineonSpice reference



Text field

The text field at the top of the dialog lets you enter and modify the text that you want to add to the schematic.

Rotate

The orientation that the text has in the schematic.

Align

The alignment of the text.

Color

Whether the text has a background.

If **Background** is checked, the text is displayed on a gray background. Otherwise, a transparent background is used.

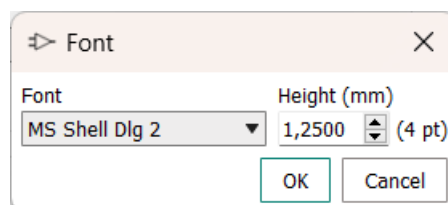
Font

The font in which the text is shown in the symbol.

The menu at the right of the dialog section lets you modify the font:

Choose Font

Opens the **Font** dialog:



The **Font** menu lets you select the desired font from the fonts installed on your computer. The **Height** field lets you set the desired font size.

Clicking **OK** applies the current setting to the **Text Attributes** dialog and closes the **Font** dialog.

Clicking **Cancel** closes the **Font** dialog without making any changes in the **Text Attributes** dialog.

Use Default Font

Sets the font information to the default font.

OK

Lets you place the specified text.

4 InfineonSpice reference

Cancel

Closes the dialog and chooses the select tool (as if you had chosen **Select** from the **Place** menu).

4.2.5 Place menu (symbol tab)

The **Place** menu of a symbol tab lets you put graphic elements into a symbol.

(For information on the **Place** menu in schematic tabs, see [Place menu \(schematics tab\)](#).)

Choosing a command from the menu selects the corresponding graphic tool with which you can draw the desired shape in the symbol view. Drawing tools stay selected until you choose a different tool or press the Escape key.

The **Pin**, **Arrow**, **Subst. Field**, and **Text** commands work in a similar way but first open a dialog in which you can configure the element to be placed.

Select

Switches to the select tool, which lets you select elements in the symbol graphic.

Pin

Lets you place grounds in the schematic.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Line

Lets you place resistors in the schematic using a symbol that is common in the EU.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Ellipse

Lets you place grounds in the schematic using a symbol that is common in the US.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Rectangle

Lets you place capacitors in the schematic.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Arc

Lets you place inductors in the schematic.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Polyline

Lets you place DC current sources in the schematic.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Polygon

Lets you place DC voltage sources in the schematic.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Chord

Lets you place diodes in the schematic.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Pie

Lets you place narrow wires in the schematic.

Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Arrow

Lets you place wide wires in the schematic.

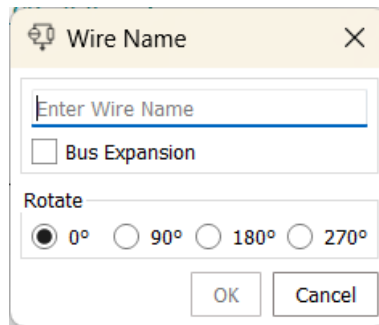
Choose a different command from the **Place** menu or press the Escape key to stop placing grounds.

Subst. Field

4 InfineonSpice reference

Lets you name wires in the schematic.

Choosing the command opens the **Wire Name** dialog:



Enter Wire Name

The name that you want to give the wire.

Bus Expansion

Rotate

The orientation of the text in the schematic graphic.

OK

Clicking **OK** closes the dialog and turns the mouse pointer into a representation of the wire name. Clicking a wire applies the name to it.

Choose a different command from the **Place** menu or press the Escape key to stop naming wires.

Cancel

Clicking **Cancel** closes the dialog and chooses the select tool (as if you had chosen **Select** from the **Place** menu).

Text

Lets you place solder dots in the schematic.

Choose a different command from the **Place** menu or press the Escape key to stop placing solder dots.

Selection Box

Lets you place lines in the schematic. Click once for placing the start of the line, and click again to place the end.

Choose a different command from the **Place** menu or press the Escape key to stop placing lines.

4.2.6 Options menu

The Options menu lets you configure the look and operation of .InfineonSpice

Shortcuts

Opens the **Keyboard Shortcuts** dialog in which you can customize the key combinations for triggering InfineonSpice functions.

For details, see [Keyboard Shortcuts dialog](#).

Display

Opens the **Display Options** dialog, in which you can configure numerous aspects of how you interact with InfineonSpice.

For details, see [Display Options dialog](#).

Default

This submenu contains a command to specify default document settings.

Schematic Template

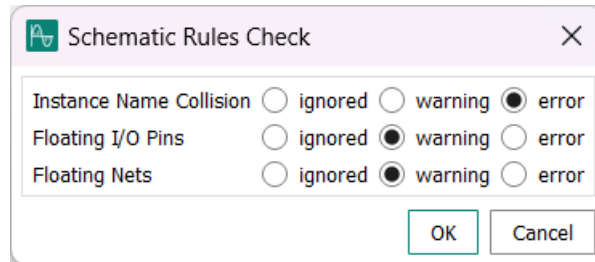
4 InfineonSpice reference

Opens the **Schematic Template** dialog, in which you can configure the dimensions of new schematic documents and the frame around the schematic.

This dialog is similar to the [Schematic Page Setup dialog](#), except that it does not include the **Apply** button. However, any settings you make in the **Schematic Template** dialog apply to future new schematic documents instead of existing ones.

Check Rules

Opens the **Schematic Rules Check** dialog, in which you can configure how certain kinds of issues in schematics are handled.



You can configure the handling of three types of issues:

Instance name collision

An instance name collision occurs when multiple components of a schematic have the same name in the schematic. This makes simulating the schematic unreliable or impossible.

Floating I/O pins

Floating I/O pins are input or output pins of a schematic that are not connected to any schematic components.

Floating nets

Floating I/O pins are wires of a schematic that are not connected to any schematic components.

For each issue, you can choose between three different handling options:

Ignored

The issue is not reported. Note that simulation results may be incorrect or misleading when any of these issues are ignored.

Warning

A message about the issue is displayed when the schematic is checked, but you can still start a simulation.

Error

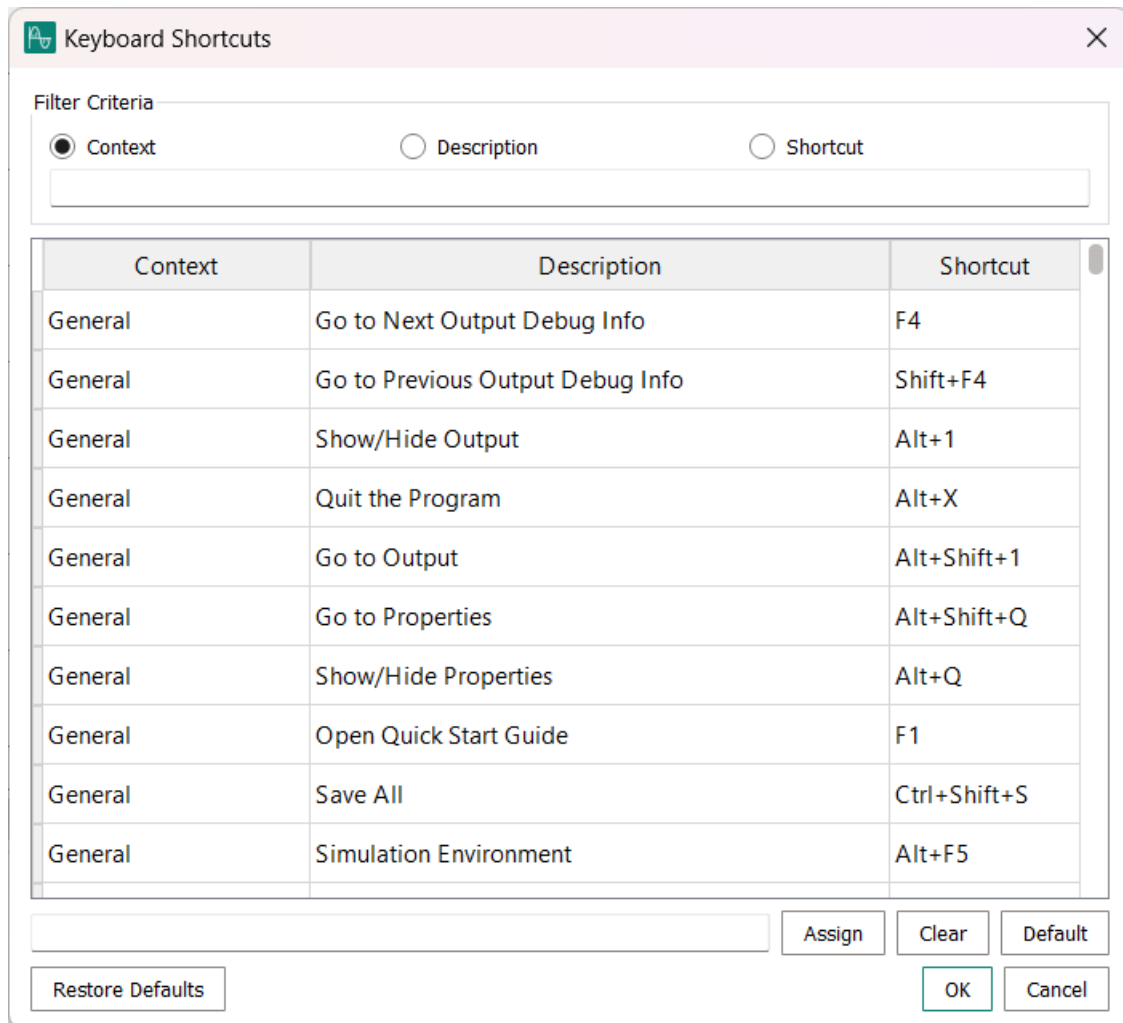
A message about the issue is displayed when the schematic is checked and the simulation is not started.

4.2.6.1 Keyboard Shortcuts dialog

In this dialog, you can configure the key combinations that trigger various functions of InfineonSpice.

The dialog is opened by choosing **Options > Shortcuts**.

4 InfineonSpice reference



Filter Criteria

The **Context**, **Description**, and **Shortcut** options specify one of the columns of the table.

When you enter text into the text field in the lower half of this section, only shortcuts that contain this text in the specified column are displayed in the table. All other shortcuts are hidden.

Main table

The main table in the dialog displays all available actions to which keyboard shortcuts can be assigned in three columns:

Context

The part of the application in which the action can be used.

Description

A brief explanation of what the action does.

Shortcut

The key combination assigned to the action, if any. If this field is empty, no key combination is currently assigned.

Text field below the table

This field lets you specify the shortcut for the action selected in the table.

To enter a shortcut, click in the field and press the desired keys together.

Assign

Assigns the key combination specified in the text field to the selected action.

4 InfineonSpice reference

Clear

Removes the key combination from the selected action.

Default

Assigns the default key combination to the selected action.

Restore Defaults

Assigns the default key combinations to all actions.

Note that the defaults are also assigned to actions that are not currently displayed in the table.

OK

Closes the dialog and sets the shortcuts of all actions to the key combination configured in the dialog.

Cancel

Closes the dialog without changing any key combinations.

4.2.6.2 Display Options dialog

The Display Options dialog lets you specify numerous aspects of how you interact with InfiniteSpice.

The dialog is opened by choosing **Options > Shortcuts**.

It contains multiple tabs for configuring different parts of the interaction, which are described below.

In addition, it contains buttons that are the same in every tab:

Default

Sets all options in the current tab to their default settings. Options in other tabs are unaffected.

OK

Closes the dialog and applies the settings that were made.

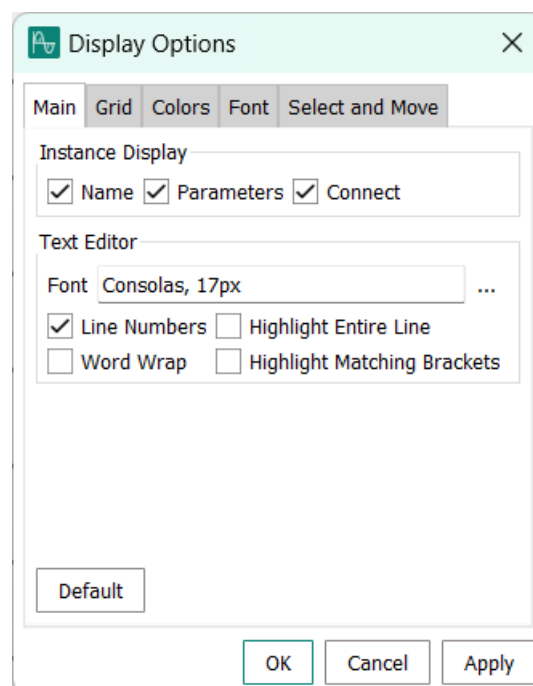
Cancel

Closes the dialog without applying any settings,

Apply

Applies the settings that were made in the dialog and keeps the dialog open.

Main tab



4 InfineonSpice reference

Instance Display

The information that is displayed alongside components in a schematic:

Name

The name of the component instance in the schematic.

Parameters

The parameters that have been set for this instance.

Connect

(Currently under review.)

Text Editor

Settings for the display and behavior of text-editing tabs, such as library tabs.

Font

The font used to display text in text-editing tabs.

Clicking the ellipsis (...) at the right of the section opens a font picker. For details, see [Select Font dialog](#).

Line Numbers

Whether line numbers are displayed in text-editing tabs.

Word Wrap

Whether lines of text wrap around when they reach the edge of the tab or continue beyond the edge (requiring scrolling to see all of such a line).

Highlight Entire Line

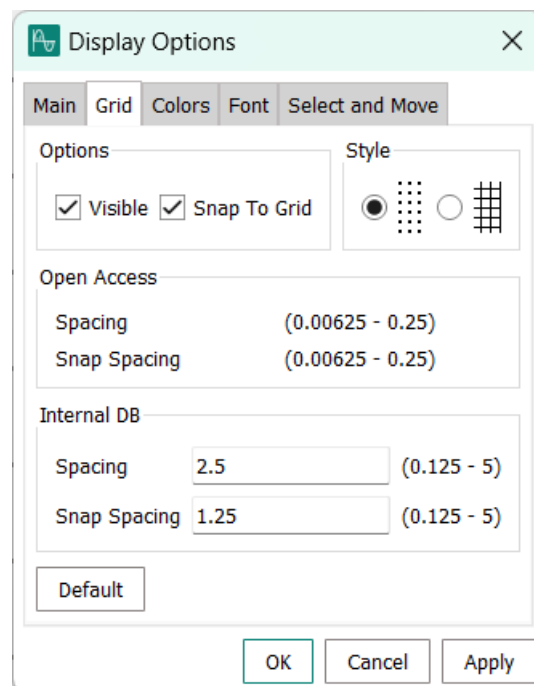
Whether clicking in a text-editing tab highlights the entire line.

Highlight Matching Brackets

Whether selecting a bracket or parenthesis of a pair – () { } [] – in text-editing tabs highlights the other bracket in the pair.

Grid tab

Settings for the grid to which you can align placed components in a schematic.



4 InfineonSpice reference

Options

Basic settings for the grid:

Visible

Whether the grid is displayed behind the components in the schematic view.

Snap to Grid

Whether components that are placed or move are automatically aligned with the grid.

Style

The graphical display of the grid.

You can choose between dots (which mark the intersection of the grid lines) or full grid lines.

This setting only makes a difference when the grid is visible.

Open Access

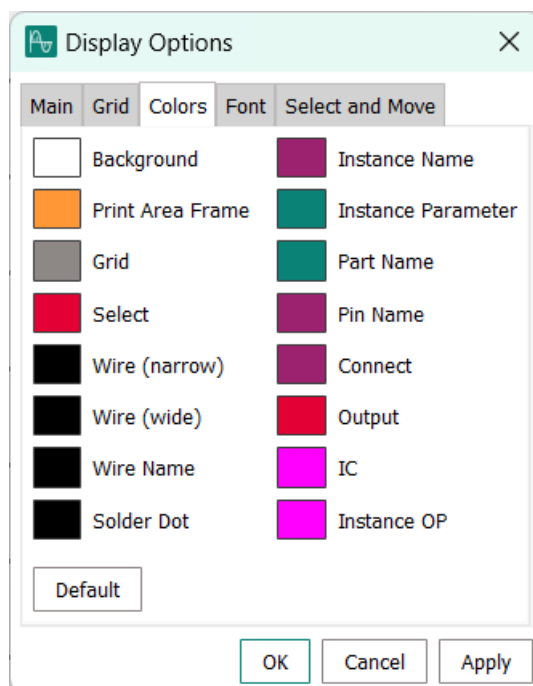
(Currently under review.)

Internal DB

(Currently under review.)

Colors tab

Lets you specify colors for various kinds of elements in schematics.

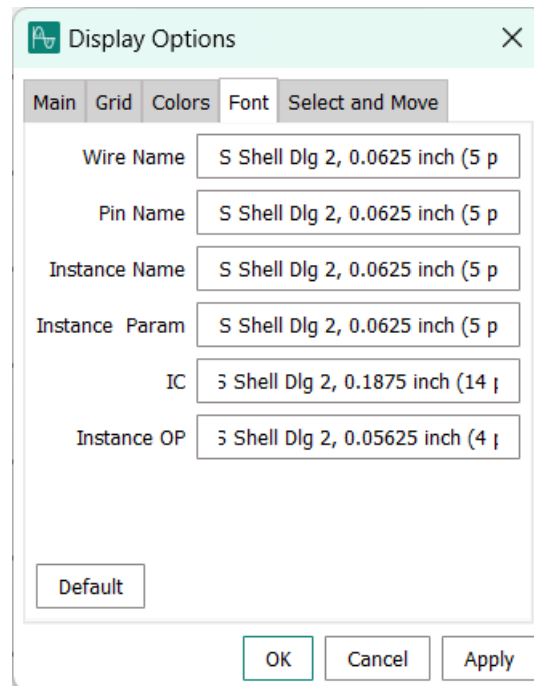


Clicking a swatch opens the [Set Net Color](#) dialog, with a modified title to indicate the element the color of which you are setting.

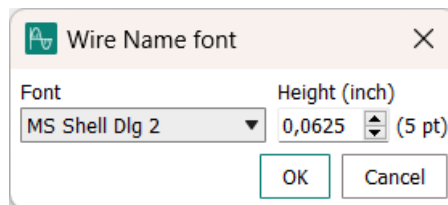
Font tab

Lets you specify the font used to display labels of elements of a schematic.

4 InfineonSpice reference



Clicking any of the text fields opens the Font dialog (with the name reflecting the type of label you are configuring):

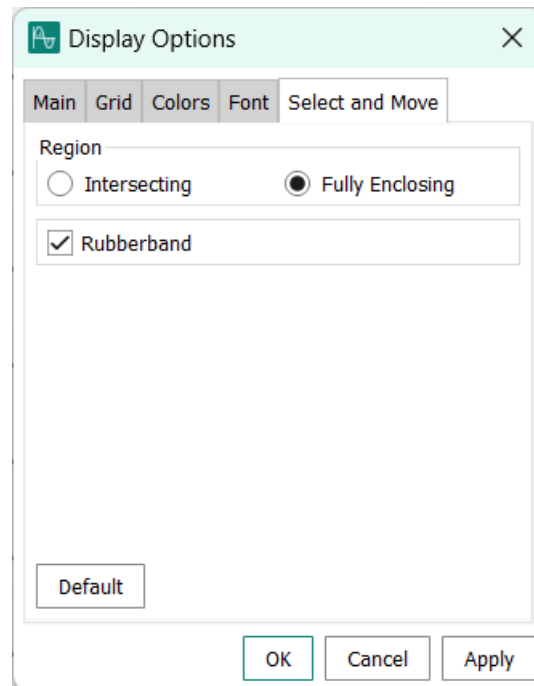


The **Font** menu lets you select the desired font from the fonts installed on your computer. The **Height** field lets you set the desired font size.

Clicking **OK** applies the current setting to the specified component and closes the Font dialog. Clicking **Cancel** closes the Font dialog without making any changes.

Select and Move

Lets you specify how click-dragging to draw a selection rectangle around elements in schematic or symbol tabs behaves.



Region

Which elements are selected:

Intersecting

An element is selected if any part of it lies inside the selection rectangle.

Fully Enclosing

An element is selected only if it is completely surrounded by the selection rectangle and no part of it lies outside the rectangle.

Rubberband

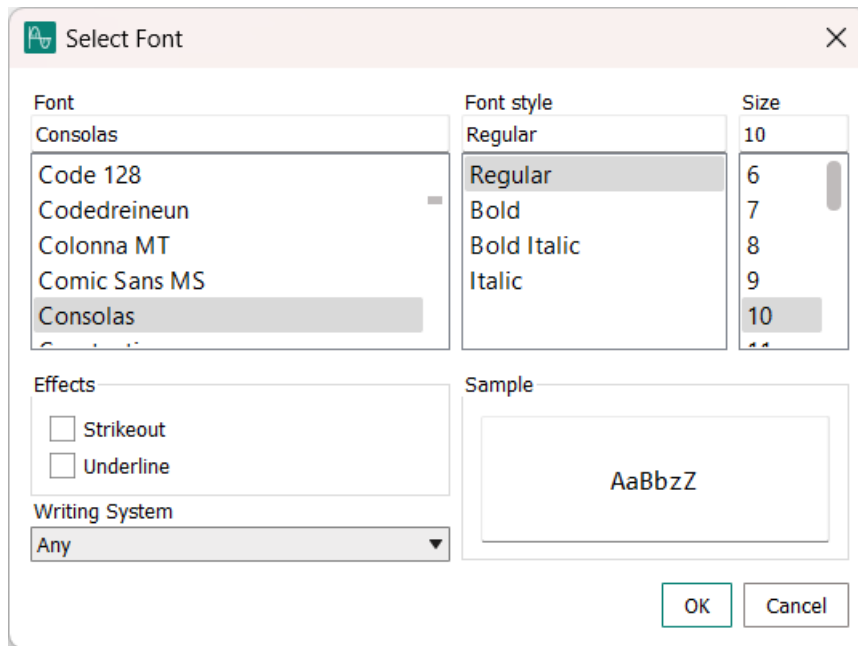
(Currently under review.)

4.2.6.3 Select Font dialog

The Select Font dialog lets you choose a font installed on the system , a style, and a size.

The dialog is opened by choosing **Options > Display** and clicking the ellipsis (...) button in the **Main** tab.

4 InfineonSpice reference



Font

All typefaces that are available on your computer that support the selected writing system.

Font Style

All styles in which the selected font is available on your computer.

Size

The size (in points) in which you want to use the font.

You can either pick a predefined size or enter the desired size into the text field above the list.

Effects

Any visual effects that should be applied to the selected font.

Writing System

The writing system (such as Latin, the standard system used in most of Europe and America, Cyrillic or Chinese) that the selected font must support.

The **Font** list displays only fonts that contain the symbols for the selected writing system.

Sample

Sample text using the settings made in the dialog.

OK

Closes the dialog and applies the settings.

Cancel

Closes the dialog without applying the settings.

4.2.7 Window menu

The **Window** menu lets you switch tabs and close them.

The menu contains four fixed commands, which are described below. At the bottom of the menu, all open tabs are listed. Choosing one of them activates the corresponding tab.

Close Tab

Closes the frontmost tab.

Close All Tabs

Closes all open tabs (but not any views).

4 InfineonSpice reference

Next Tab

Makes the next tab active.

Previous Tab

Makes the previous tab active.

4.2.8 Help menu

The **Help** menu contains information on the application and links to helpful resources in the Internet.

Welcome

Opens the **Welcome** tab in the main part of the application window.

User Guide

Opens the InfineonSpice user guide in a browser window.

Ask the Community

Opens a browser window showing the InfineonSpice discussion forums on the Infineon website.

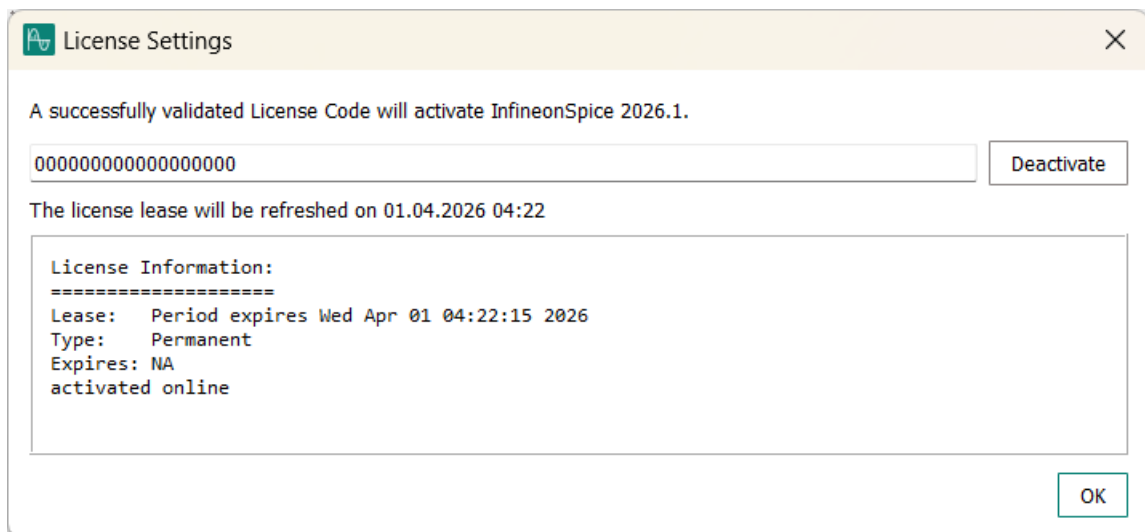
Check for Updates

Opens the InfineonSpice download page on the Infineon website.

If there are newer releases than the one you are using, you can download them from here.

License Settings

Opens the License Settings dialog.



License code

The text field near the top of the dialog displays the current license code, if any.

Entering a different code into this field lets you switch to another license by clicking **Validate**.

Validate

If you have modified the content of the license code field, the **Validate** button is shown. Clicking it validates the license and lets you use InfineonSpice according of the parameters of the license.

Deactivate

If a license is activated and you have not modified the content of the license code field since opening the **License Settings** dialog, the **Deactivate** button is shown. Clicking it deactivates the current license and immediately terminates InfineonSpice when the **License Settings** dialog is closed.

License information field

This field displays various pieces of information about the currently active license.

OK

4 InfineonSpice reference

Closes the dialog.

Attention: *If you have deactivated the license and not activated a new one, clicking OK immediately terminates InfineonSpice. Any unsaved changes are lost.*

About

Displays version, copyright, and licensing information.

4.3 Windows and views

The center of the window is the main work area. It contains a tabbed view for each object on which you are working (such as schematics or models).

Each of these views can be extracted from the main window and can stand on its own by clicking



and dragging the view to the desired

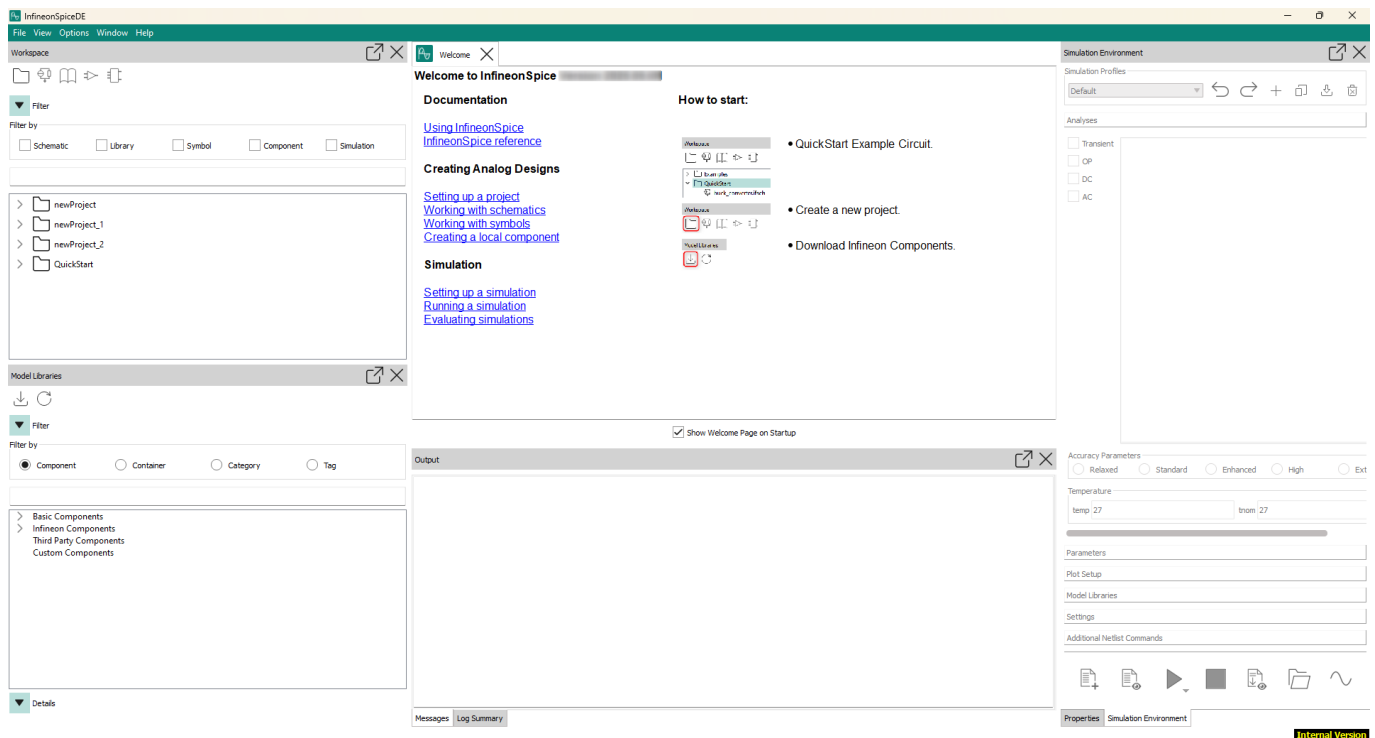
position.

To move a view to another place grab it on the title bar and move it to the favored place. Blue areas indicate the dropping point of the view.

One or more views can also be closed by clicking



If you want to return to the default layout, you can do so by clicking **View > Reset View Layout**.



At the bottom of the window, the **Output** view provides feedback from the system. Refer to [Chapter 2.1](#) in the Getting started guide.

4 InfineonSpice reference

4.3.1 Work area

The work area is where you display and edit schematics, symbols, and other documents. If no document is open, it displays helpful links and tips.

4.3.2 Workspace view

The **Workspace** view at the top left of the design environment contains all folders, projects, schematics, libraries, symbols and components, netlists and logs. This is the storage or directory from which to navigate.

Workspace icons











New Project

New Schematic
File name suffix is .ifsch

New Library
File name suffix is .lib

New Symbol

New Local Component

create:



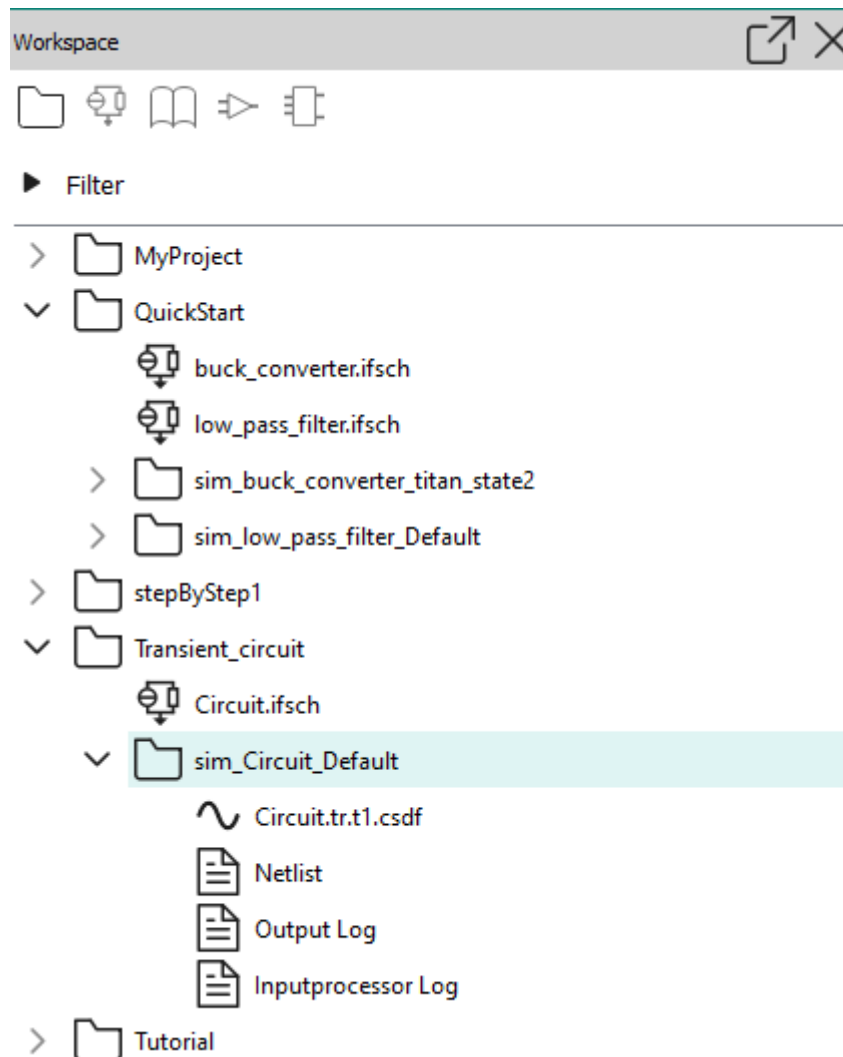




Workspace

The workspace contains for example

4 InfineonSpice reference



Project logs

Each project also contains a folder that includes the waveform, netlist, output log and input processor. The netlists and waveform can also be found in the **Simulation Environment**. These are described in [Simulation Environment view](#)

If you double-click a log file in the **Workspace** view, it opens in the **Output** view's **Log Summary** tab. You can filter the tab's content by message category.

The **Output** view's **Messages** tab contains a list of status messages that were generated while executing the simulation. At the bottom of the window, the **Output** view provides feedback from the system.

The Output gives messages as well as Log Summary.

The messages are given in the form of text.

The log summary is given in the form of a table that includes **Info**, **Warning**, **Error**, **Severe**, **Fatal**.

Click on any one of the headings to select or deselect.

4 InfineonSpice reference

Output				
			Info (24)	Warning (4)
			Error (2)	Severe (0)
			Fatal (0)	
Line	Severity	Description		
13	Info	Preprocessing passed. 0 subcircuit definitions found.		
21	Info	The compatibility mode has been set to COMPAT=PSPICE by...		
41	Warning	Message groups no longer supported, ignored.		
52	Info	The compatibility mode has been set to COMPAT=TITAN by .COMOPT...		

Line: The number refers to the line in the **Inputprocessor Log** for the simulation.

Info: message is treated as information.

Warning: message is treated as a warning.

Error: message is treated as an error, continue with next analysis.

Severe: message is treated as a severe error, continue with next alteration (if there is one)

Fatal: message is treated as a fatal error, abort simulation

4.3.2.1 Project menu

Right-click on a project icon,  which is in the vertical list, and you will get a menu with the following commands from which you can choose:

	New Schematic...	
	New Library...	
	New Symbol...	
	New Local Component...	
	Show in File Explorer...	
	Copy...	
	Export...	
	Rename...	F2
	Remove	Alt+Del

New Schematic

Create a new schematic in the work area by right clicking on the project and select **New Schematic**

New Library

Create a new library in the work area by right clicking on the project and select **New Library**

New Symbol

Create a new symbol in the work area by right clicking on the project and select **New Symbol**

New Local Component

You can only create a new component when one Library file containing one model is included.

Show in File Explorer

By right-clicking on the schematic, you can locate the directory to which the schematic is saved.

Copy

You can copy the project and choose the location to which it should be copied. If you do not give it a new name, it will rename itself by adding a digit to the original name.

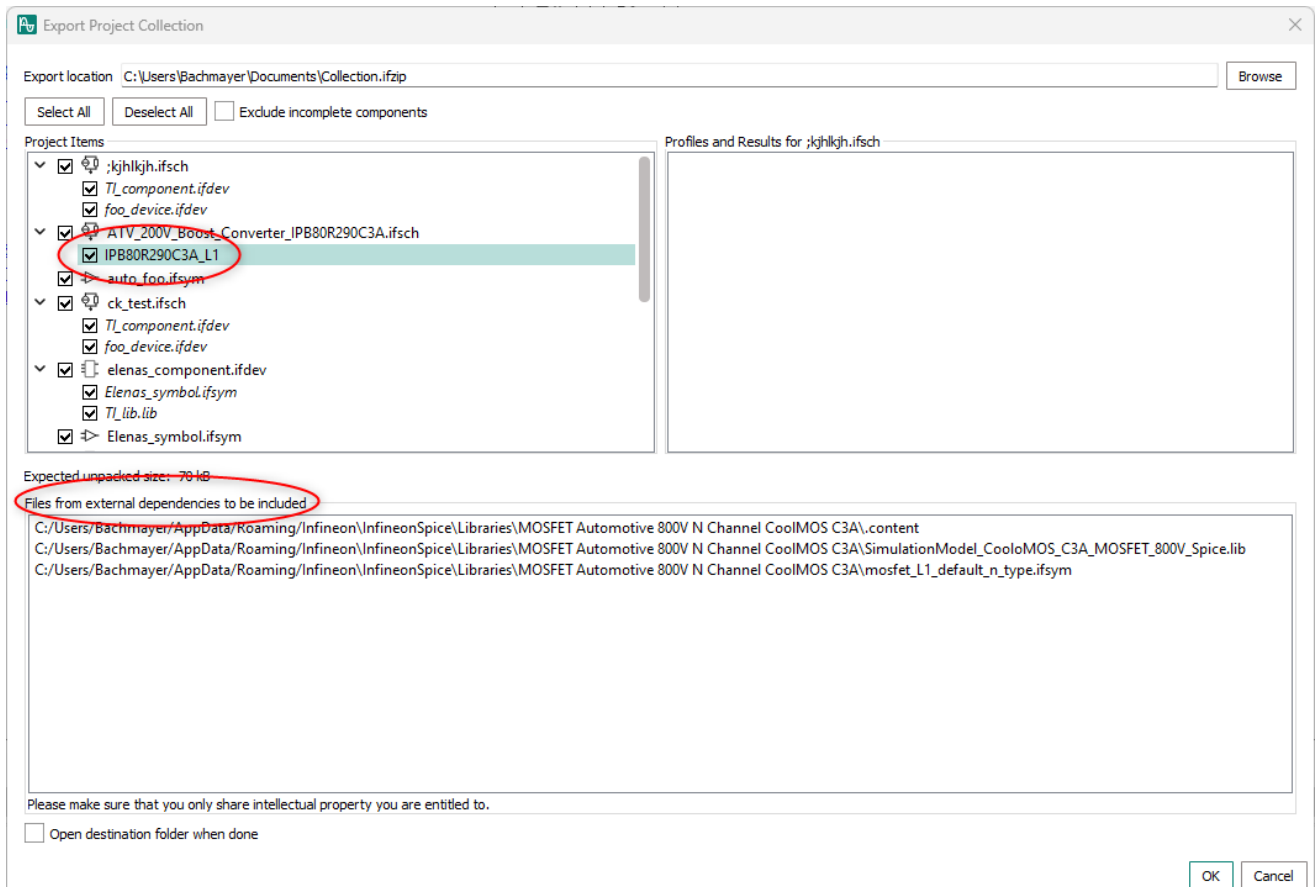
4 InfineonSpice reference

Export

You can export/save the project and choose the location to which it should be exported. If you do not give it a new name, it will rename itself by adding a digit to the original name.

When you click **Export**, a window opens with several options. The name of the folder appears on the top left, in this example it shows **Export** which is the name of the command, **Project** is the starting point of where you selected this command. **Collection** is the name of your project.

Within this folder, you will have a number of components and symbols, for example, that you potentially want to export.



Within the window, you give your **Export location** of your choice. You can **Browse** for the location of your choice. Secondly, you may **Select All** or **Deselect All** or only mark the boxes of the selection of your choice, manually. You have the option also to **Exclude incomplete components** by checking the box. The window contains **Project Items**, **Profiles and Results for <Name of schematic>**.

The **Expected unpacked size: xx kB** is shown.

A window **List of files from external dependencies that will be included:** shows all external files that are dependent on your export. In this event, be aware that these files cannot be edited.

Bottommost on the left hand side, you are reminded not to share intellectual property that you are not entitled to. Underneath this statement, you have the choice to **Open destination folder when done** by checking the box. Click **OK** to proceed or **Cancel** to abort.

Rename

You can rename a file.

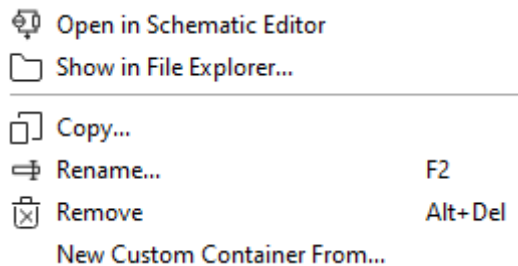
Remove

Delete the file from view.

4 InfineonSpice reference

4.3.2.2 Schematic menu

Right-click on a schematic icon , which is in the vertical list, and you will get a menu with the following commands from which you can choose:



Open in Schematic Editor

Display a schematic in the work area by right clicking on the schematic and select **Open in Schematic Editor**

Alternatively, you can also double click on a schematic from the **Workspace** to display in the work area.

Show in File Explorer

By right-clicking on the schematic, you can locate in which directory this schematic is saved.

Copy

You can copy the schematic and choose the location to which it should be copied. If you do not give it a new name, it will rename itself by adding a digit to the original name.

Rename

You can rename a file.

Remove

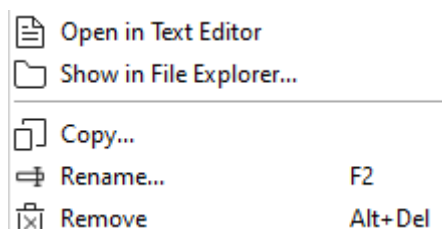
Delete the file from view.

New Custom Container From

You can place all of your schematics in a container of your choice. When you do this, you have to name of the container. You will be asked if the dependencies should also be copied, if there are any to other existing containers. Find your new containers in the **Model Libraries > Custom Components**

4.3.2.3 Library menu

Right-click on a library icon , which is in the vertical list, and you will get a menu with the following commands from which you can choose:



Open in Text Editor

A text editor opens where you can edit your library.

Show in File Explorer

By right-clicking on the library, you can locate in which directory this library is saved.

Copy

4 InfineonSpice reference

You can copy the schematic and choose the location to which it should be copied. If you do not give it a new name, it will rename itself by adding a digit to the original name.

Rename

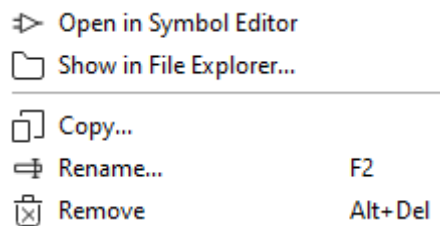
You can rename a file.

Remove

Delete the file from view.

4.3.2.4 Symbol menu

Right-click on a symbol icon , which is in the vertical list, if your project contains an individualized symbol and you will get a menu with the following commands from which you can choose:



Open in Symbol Editor

The work area opens and you can edit the symbol.

Show in File Explorer

By right-clicking on the library, you can locate in which directory this library is saved.

Copy

You can copy the symbol and choose the location to which it should be copied. If you do not give it a new name, it will rename itself by adding a digit to the original name.

Rename

You can rename a file.

Remove

Delete the file from view.

4.3.3 Model Libraries view

The **Model Libraries** view presents all available standard components that have been made available by Infineon and other sources and that you can use in your circuits.

Each device typically would contain a set of its own symbols. These symbols, components and other schematic data are contained in a library. When you simulate your environment with the use of a specific device, then download a library and the various elements in the library can be arranged according to your application for the simulation. This saves looking for the items required in the vast ocean of elements because you can find the required subset in the library.

Download Manager

for information on using the Download manager, see Managing Infineon and third-party components.

Download the required library by means of the **Download Manager** by clicking the icon .

Filter the list to ease your search by either

- **Component**
- **Container**

4 InfineonSpice reference

- **Category**
- **Tag**

A window pops up and this can then be sorted for **Components, Circuits, Automotive only**. Select the required library and download.

Basic Components

You have a choice of various standard circuit components to drag and drop onto your schematic. Basic components are listed in the Model library as well as on the task bar in the work area as follows:

Capacitor

Drag and drop onto work area, press the Esc key to complete. You can also right-click to trigger the **Place** command. Move your mouse onto the work area and place your component in the desired position.

Controlled sources

Drag and drop onto work area, press the Esc key to complete. You can also right-click to trigger the **Place** command. Move your mouse onto the work area and place your component in the desired position. You can choose from a number of controlled sources as listed below:

- Voltage-controlled current source (vccs)
- Voltage-controlled voltage source (vcvs)
- Current-controlled current source (cccs)
- Current-controlled voltage source (ccvs)

Current Source

Drag and drop onto work area, press the Esc key. You can also right-click to trigger the **Place** command. Move your mouse onto the work area and place your component in the desired position. You can choose from a number of current sources as listed below:

4 InfineonSpice reference



DC



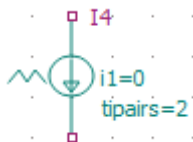
Variable/Behavioral



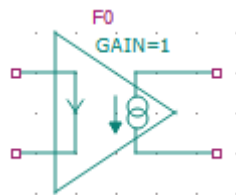
Pulsed



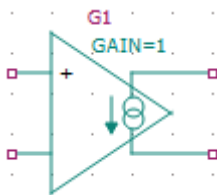
Sinusoidal



PWL



Current-controlled current source (cccs)



Voltage-controlled current source (vccs)

Diodes

You have a number of diodes from which to choose depicted in the menu.

- **Ideal Diode**
- **Zener Diode**
- **Schottky Diode**
- **Model based Diode**

Ground

Drag and drop onto work area, press the Esc key to complete. You can also right-click to trigger the **Place** command. Move your mouse onto the work area and place your component in the desired position.

VDD Jumper

Drag and drop onto work area, press the Esc key to complete. You can also right-click to trigger the **Place** command. Move your mouse onto the work area and place your component in the desired position.

4 InfineonSpice reference

Ideal Transformer

Drag and drop onto work area, press the Esc key to complete. You can also right-click to trigger the **Place** command. Move your mouse onto the work area and place your component in the desired position.

Inductance

Drag and drop onto work area, press the Esc key to complete. You can also right-click to trigger the **Place** command. Move your mouse onto the work area and place your component in the desired position.

NC

Drag and drop onto work area, press the Esc key to complete. You can also right-click to trigger the **Place** command. Move your mouse onto the work area and place your component in the desired position.

Resistor

Drag and drop onto work area, press the Esc key to complete. You can also right-click to trigger the **Place** command. Move your mouse onto the work area and place your component in the desired position.

Infineon Components

Download Infineon components from the **Download Manager**. Downloaded components appear in the categories as listed below.

These circuits are prebuilt. You need to save it with a different name because all Infineon components are read-only.

Two categories of Infineon components in the components list are:

Circuit Testbench depending what you downloaded from the **Download Manager**, can include **Power Management** and **Automotive**

Power depending what you downloaded from the **Download Manager**, can include **gate-driver-ics**, **mosfet** and **igbt**

4.3.4 Simulation Environment view

The **Simulation Environment** view provides detailed information about the setup used for simulations, which you can modify.

Simulation Profiles

Set a simulation profile in an open schematic. This can be Default or any other that has been created. The profile defines settings of your choice for electrical characteristics values in each of the components in the schematic.

Analyses

Select the desired analyses that you want the simulation to perform:

- **Transient**
- **OP** (operating points)
- **DC** (direct current)
- **AC** (alternate current)

At least one analysis must be selected.

Accuracy Parameters set for **Relaxed**, **Standard**, **Enhanced**, **High** or **Extreme** according to the strictness required for your simulation. Set the **Temperature** range required.

Do not select **Skip initial bias point computation** unless you have specific reasons to do so. See the Titan User manual in the section about UIC (Use initial condition) for details.

Parameters

Additional parameters that are not listed in the Properties for the individual components can be added here.

4 InfineonSpice reference

Plot Setup

You can select how much detail for voltages and the currents to be displayed in the plots that are generated:

- **All:** Every voltage or current, including in subcircuits
- **Top-level:** Every voltage or current in the schematic itself, but not those in subcircuits
- **Signals only:** Only voltages or currents on the specified signal

You can specify which signals to be displayed in your plot. The signals are given in a list. Right-click the list to specify your signal choice. You can

- **Delete Row(s):** Excludes signals not required for your plot
- **Select Output on Schematic:** Select the schematic for the required signals to include in your plot
- **Select Output on Schematic (differential):** Select the schematic for the required signals to include in your plot
- **Add Expression** Any expression that you wish to plot can be created

Plot Mode for your simulation, you can choose

- **No plotting**
- **All outputs in one plot**
- **Every output in separate plot**

If you deal with complex numbers, you can specify which signals to be displayed in your plot. The signals are given in a list.

Click on the **AC Modifier** and you have a number of points of analysis that can be selected on the schematic including:

- **Magnitude**, which you can select on the schematic to see magnitude of a particular element.
- **dB20**
- **Phase**
- **Real part**
- **Imaginary part**

Model Libraries

Choose the Library Path by placing the link the field **Library Paths**. A Library contains components. See Custom containers

Settings

You set **Floating Nodes** and **Relative Reference** in the Settings field.

Floating Nodes: You can add a shunt resistor and select the size.

Relative Reference: You can set how the relative reference value for the voltage/current-based integration error control is computed. The default value is 'Point Global'.

Additional Netlist Commands

Add any further commands as you wish in this window that you want in the simulation. Click on the icon



to create a netlist.

You can review the log messages generated during the simulation by clicking .

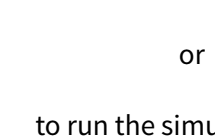
Run a simulation

Refer to Running a simulation.

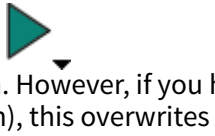
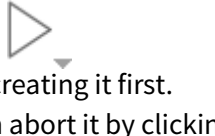
The following icons below the Simulation Environment window are shortcuts for the commands explained. Hover over them with your mouse to see what they command



4 InfineonSpice reference

Click either  or  to run the simulation.

You can switch between the two buttons by clicking the triangle in their lower-right corner and choosing the desired button from the menu that opens.

You run the simulation using the  button. This recreates the netlist and immediately runs the simulation. However, if you have already generated the netlist and edited it (as described in step Running a simulation), this overwrites your changes. In that case, use the  button which runs the simulation using the existing netlist, without recreating it first. If a simulation runs too long, you can abort it by clicking 

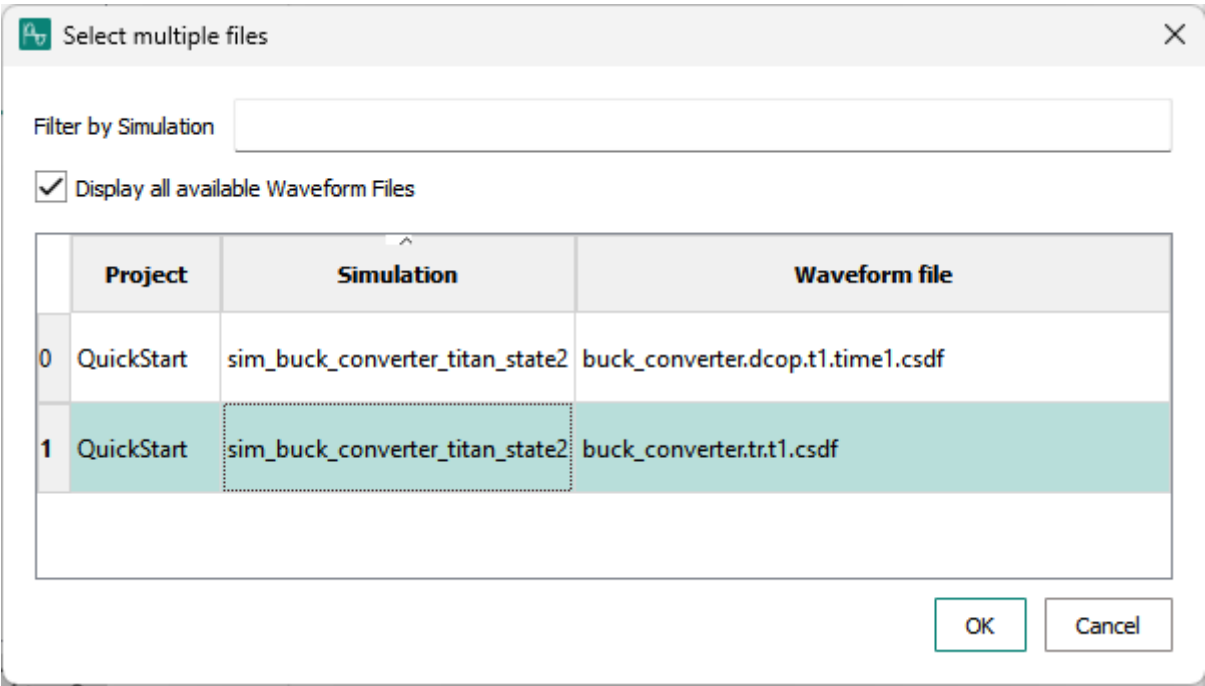
In this case, no simulation results are created and the Waveform Viewer is not opened. Review the log messages generated during the simulation by clicking 

Access the files generated by the simulation by clicking 

Review the simulation results in the Waveform Viewer. If the Waveform Viewer does not open automatically, you can open it manually by clicking 

Refer to [Workspace view](#) for information about Output **Messages** and **Log Summary**. Open the Waveform Viewer with a selection in the **Analyses section** . If no selection is made in the **Analyses section**, a table appears after clicking on 

, at which time you can then choose from what is currently available. After you make a choice of one or several files from the table as shown below, your waveform appears.



4 InfineonSpice reference

Refer to [Waveform viewer](#).

4.3.5 Properties

The **Properties** view displays detailed information about a selected component and lets you modify it.

Open the **Properties** view and fill out the necessary and desired parameters. Necessary parameters are highlighted in yellow until you have filled in a value.

The default position for **Properties** menu is under simulation on bottom right-hand corner of the InfineonSpice

4.3.6 Output

The **Output** displays all actions taken during a simulation.

Output

Output contains **Messages** and **Log Summary**

When a simulation is run, all relevant information including warnings and errors are found in this view.

4.4 Waveform viewer

The Waveform Viewer displays the graphs that the simulation you set up would produce. A number of icons also represent some of the commands available in the toolbar.

InfineonSpiceWV

After you follow the steps in [Setting up a simulation](#), you might want to run a simulation, [Running a simulation](#).

Click  and the WaveformViewer opens automatically with the relevant waves that your schematic will output.

The following menus appear on the menu bar:

- [File](#)
- [View](#)
- [Plot](#)
- [Trace](#)
- [Tools](#)
- [Help](#)

4 InfineonSpice reference

4.4.1 File

Click **File** and a list of options related to managing files, such as to open, save, print and import data and exit, appears.

File	View	Plot	Trace	Tools	Help
	Open...			Ctrl+O	
	Open with Layout...			Ctrl+Shift+O	
	Reload			Ctrl+R	
	Save			Ctrl+S	
Save As...					
Export CSV...					
Print Preview...					
	Print...			Ctrl+P	
	Print Setup...				
Export as SVG...					
	Copy to Clipboard...				
Close No Save					
Close				Alt+X	

Open

Click **File > Open** or select



location.

to open a saved file in your selected

Open with Layout

Open with Layout means that not only do you get the waveform but also get all of the toolbars including all of the icons, and **Output Variables**

Reset

Click **Reset** or



layout back to the default view. If you zoomed into the graph or rearranged the windows, all of these actions will be reversed to the default.

to change your

Save

Secure all changes by clicking **Save** or



.

Save As

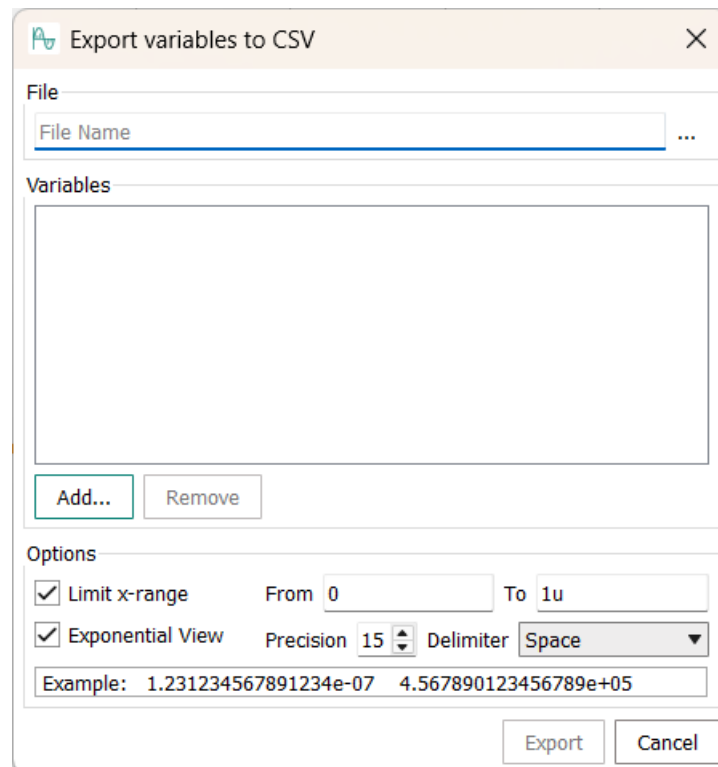
Saves the output of this simulation with a name of your choice.

Export CSV

Click **Export CSV**.

A window, **Export variables to CSV** opens. **File > File Name** appears and you give the file a name or browse to overwrite an existing file.

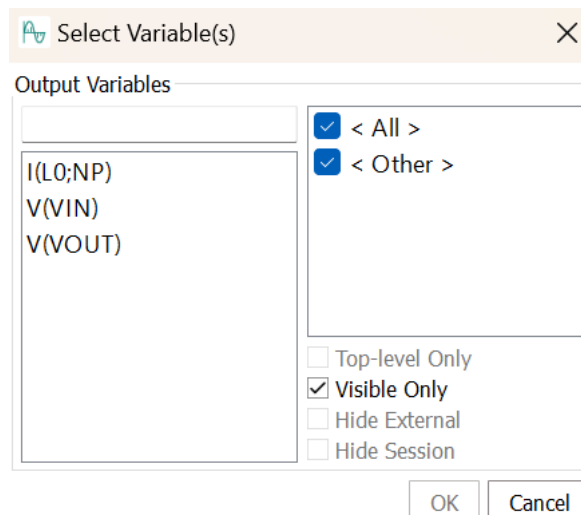
4 InfineonSpice reference



Click **Add** and a **Select Variable(s)** window open. Here you choose from the available list of variables that are included in these plots. **Remove** allows the removal of your variables in case you change your mind.

Within the **Select Variable(s)** window, **Output Variables** contains a search field where you type the variable you require and the rest of the list empties out if it does not match your search. Remaining in the list is the item for which you have searched. On the bottom right of the **Select Variable(s)** window, you can select variables according to their position in case you have several plots open.

Click **OK** to proceed with your selection. Click **Cancel** to abort **Select Variable(s)**.

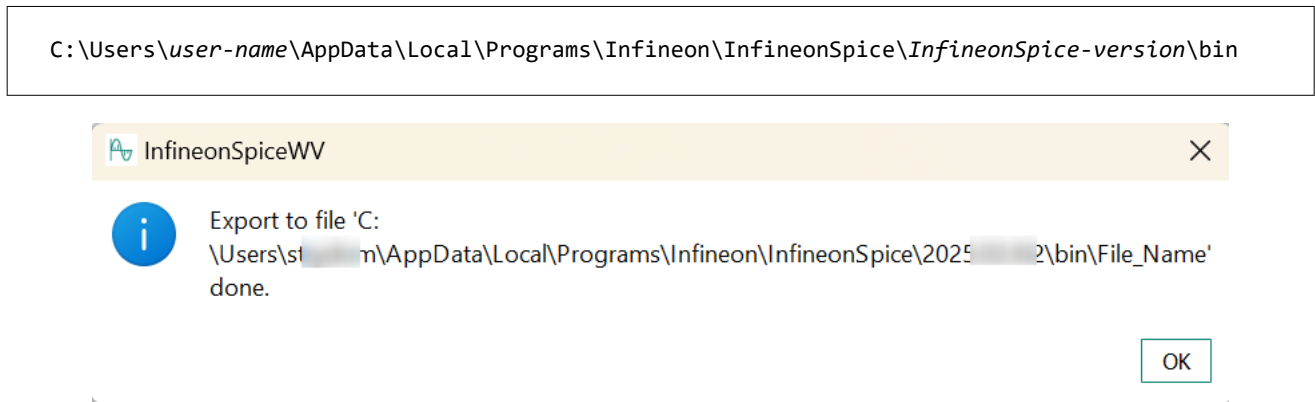


If you clicked **OK** you will have the variables for export in the **Variables** window.

The **Limit x-range** option lets you restrict the export to a specific range. The **Exponential view Precision** defines how many decimal places after the decimal point anywhere between 0 and 15. You can set the **Delimiter** which is what will separate the various data units.

4 InfineonSpice reference

Once you have completed all the settings, click **Export**. A window opens indicating the folder in which you can find your file:



Alternatively, click **Cancel** to abort the process.

Print Preview

You can view what you would obtain if you printed the waveform before printing.

Print

Click **Print** and a window opens where you may choose how many copies you wish. You can also choose to **Collate**, specifically arrange printed documents in a specific order, assembling each set of pages in the correct sequence or **Print First Page First** or **Print Last Page First**. Click **Print** and an explorer window opens where you then select the location of the printout in PDF format.

Print Setup

Set all desired settings for the Printer, Orientation, Copies, Colour mode, Print to file → Slide button Off or On and Pages → All pages or Custom range.

Export as SVG

Export the diagram as a standard vector graphic (SVG) instead of PDF which is the output of the **Print** menu.

Copy to Clipboard

If you would like to paste the waveform to another any other, for example, Microsoft Office application, click on **Copy to Clipboard**

Close No Save

Click on **Close No Save** and you close the Waveform Viewer window. Nothing is saved.

Close

Select **Close** and a window opens where you can choose to save changes but only if you made any edits. You can also check the box for whatever selection you choose as your default.

4.4.2 View

View refers to a specific arrangement or presentation of the content within InfineonSpice. The information is displayed in a list format. Depending on what kind of content is displayed in the frontmost tab, different commands are available for **View**

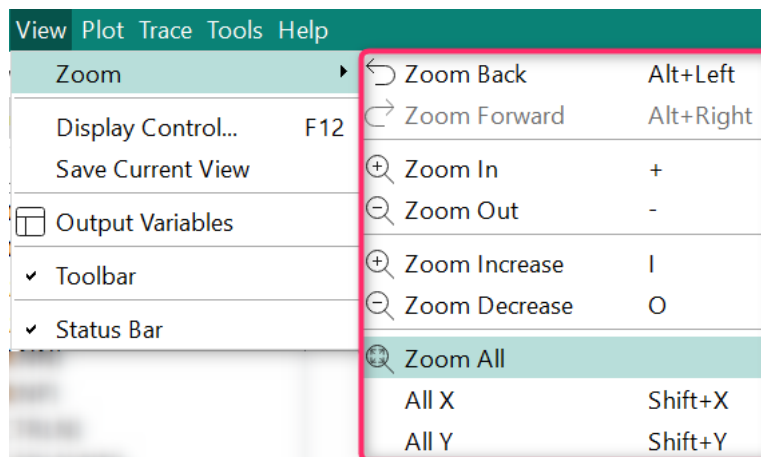
The icons in the relevant menus can also be found on the toolbar of the Waveform Viewer workspace and can have the same purpose as the menu.



Zoom

Click on **Zoom** and a number of zooming options appear.

4 InfineonSpice reference



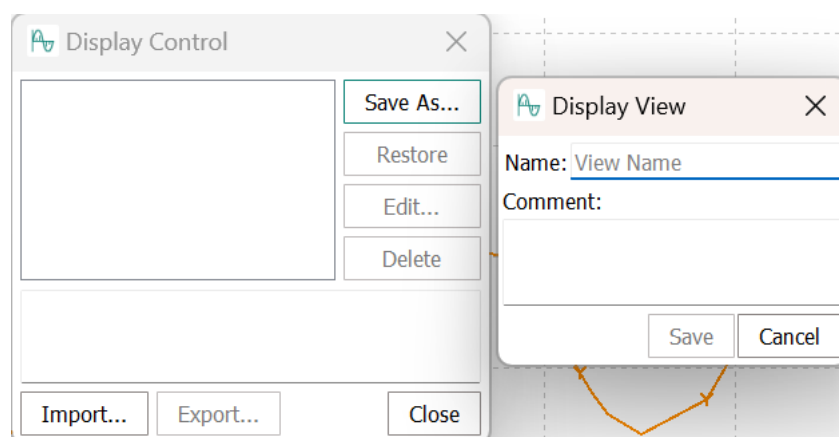
Zoom Back reverts the zoom back to the original position. **Zoom Back** can only be used once you triggered a zoom command.

Zoom Forward reverts the **Zoom Back**.

Zoom In to see more detail.

Zoom Out to see the bigger picture

Display Control



Save Current View

Output Variables

Click on **Output Variables** to display the output variables menu or hide it.

Toolbar

Click on **Toolbar** to display the Toolbar or hide it.

Status Bar

Click on **Status Bar** to display the status bar on the page bottom or hide it.

4.4.3 Plot

Outputs are the signals that are plotted in the Waveform Viewer after the simulation. You can add more outputs after running the simulation, but specifying them upfront helps you see the important results immediately.

The following choices come up when you click **Plot**

4 InfineonSpice reference

Plot	Trace	Tools	Help
	Add Y Axis		Ctrl+Y
	Delete Y Axis		Ctrl+Shift+Y
	Auto Scale Y Axis		
	Add Plot		Ctrl+L
	Delete Plot		Ctrl+Shift+L
	Move Plot Up		Ctrl+Up
	Move Plot Down		Ctrl+Down
	Unsynchronize X Axis		Shift+Pause
	Combine Traces in Plots		Ctrl++
	Scatter Traces in Plots		Ctrl+-
	Scatter Traces and Zoom		Ctrl+*
	Resize Plots Manually		
	Adjust Size		▶
	Labels		▶

Add Y Axis

If you wish to add more variables to the same plot, you can add another Y Axis.

Delete Y Axis

Any additional y-axes added can be deleted with the **Delete Y Axis** command

Auto Scale Y Axis

Add Plot

Double click on any of the variables in the **Output Variables** window and the trace for the variable appears in the plot.

In addition, check the box **in new X Plot** and the new trace is depicted in a plot of its own and not combined with other plots.

Delete Plot

Move Plot Up/Move Plot Down

Button **Move Plot Up** or **Move Plot Down** the plot will move up or down independent of existing x-axes groups

If an x-axis group is empty it will be removed completely.

Move Plot Up: The top most plot appears at the bottom

Move Plot Down: The bottom most plot appears at the top

Unsynchronize X Axis

Use this command or select , to be able to adjust width of individual axes. If synchronized, all widths have to be the same of individual plots.

Combine Traces in Plots

The data series, depicted in the plots, if there is more than one, can be displayed in one x-axis and y-axis altogether.

Scatter Traces in Plots

If you have more than one trace and the traces are combined, in one single x-axis and y-axis plot, you can scatter or split them, each in their own plot.

4 InfineonSpice reference

Scatter Traces and Zoom

If you have more than one trace and the traces are combined, in one single x-axis and y-axis plot, you can scatter or split them, each in their own plot.

Resize Plots Manually

Select this command, and as soon as you reach an x-axis, the icon,  appears. Click the icon and keep the mouse depressed while you pull the axis as high or as low as you like.

Adjust Size

Height, **Auto** and **Width** are three different actions which have different effects on the layout of plots.



Height adjusts the height of the plots within their synchronization group

Width adjusts the width of the synchronization group (if they are next to each other and not above/below)

Auto adjusts everything, so the synchronization groups and plots are all proportional

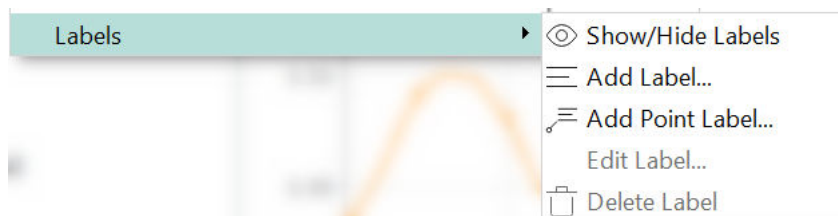
To enable the action **Width**, do the following:

1. Click on **Tools > Options**, uncheck **Horiz. Unsynchronize X-Axis**. Click **OK**
2. Add at least one other plot. uncheck **Horiz. Unsynchronize X-Axis**. Click **OK**
3. Now, when resizing the plots, the width of the synchronization groups can be changed

To enable **Height** there needs to be at least one synchronization group with at least two plots.

To enable **Auto** there need to be at least two plots.

Labels



Show/Hide Labels makes any labels visible or not visible. However, labels are created first by adding labels.

Select **Add Label** and a window opens where you can place your text. You can choose **Rotate** 0°, 90°, 180° or 270°. **Align** the text horizontally either **Left**, **H.Center**(horizontal center), **Right**. You can also align your text vertically being: **Left**, **V.Center**(horizontal center), **Right**. Select the bottom right block of the **Label** window to select font. The dropdown list gives a choice **Choose font** or **Use default font**. The default font is shown as MS Shell Dig 2



Output Variables

A list is here of all the possible variables that the simulation comprises.

Under the list of variables a sub menu facilitates the plot handling as well as the depiction of the variables in the box above.

4 InfineonSpice reference

☐ Top-level Only
 ☐ Hide External
☐ Visible Only
 ☐ Hide Session

☐ in new X Plot

Trace Functions ▼

Check the **Top-level Only** box to sort variables and show only the top-level variables.

When you have a number of plots and wish to see which variables are traced and remove any variables that are not included, check the box **Visible Only**

When you want to add one of the variable traces in its own plot immediately, check the box **in new X Plot**, then any new trace will not be combined with any other.

Select a variable, click **Add** and you will get your plot named under the relevant plot.

Trace Functions

Underneath the output variables, all the trace functions can be found. Mathematical, analytical and statistical functions are available for you evaluation of the trace in the plot.

The big arrow/dropdown list next to the **Trace Functions** is a shortcut to some of the mathematical and statistical functions that would be required.

☐ Top-level Only
 ☐ Hide External
☒ Visible Only
 ☐ Hide Session

☐ in new X Plot

Trace Functions

Freq Analysis... Alt+Ins

Σ Evaluate Goal Function... Shift+E

Separate Branch... Shift+S

Create Eye Diagram... Shift+D

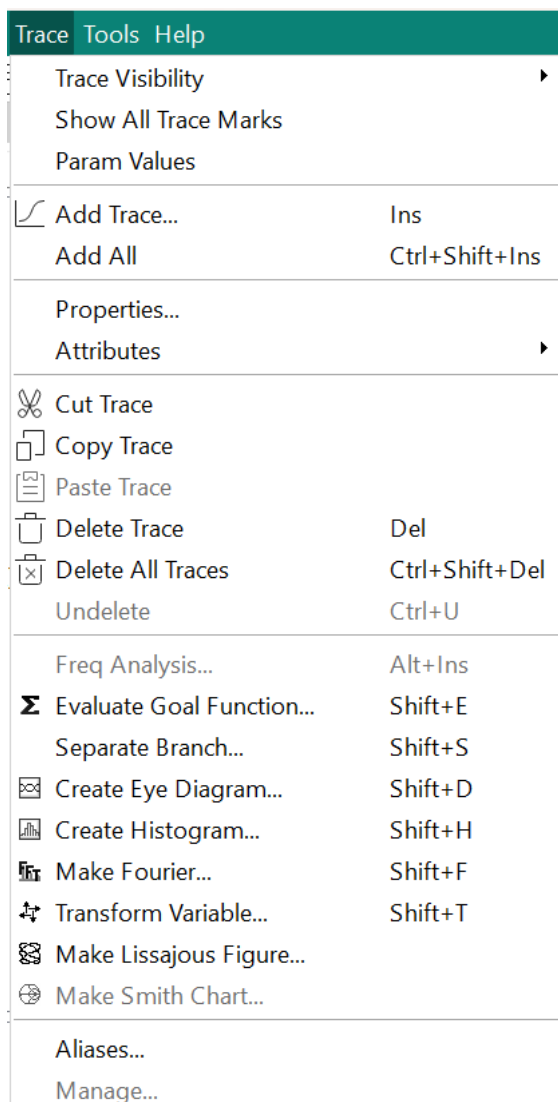
Create Histogram... Shift+H

Make Fourier... Shift+F

Transform Variable... Shift+T

4 InfineonSpice reference

4.4.4 Trace



The icons found in the menu next to the command can also be found in the tool bar. Click directly on the icon in the toolbar to select the command.

Trace Visibility

Trace Visibility menu contains a dropdown list where you can hide the lines that connect the simulation values, so you only have the trace plot marks. Alternatively, you can also choose to hide lines. Only one of the choices is possible or else the plotting of all simulation values would be invisible.

Show All Trace Marks

All simulation values are depicted on the plot. Click **Show All Trace Marks** to select and click again to deselect the command

Param Values

All parameter values are depicted on the plots. Click **Param Values** to select and click again to deselect the command

Add All

Add All triggers traces of all available output variables in the plot regardless of whether they are visible in the list or not. Note that once you have triggered **Add All**, you have to select each plot individually and then select **Delete Plot**.

Properties

4 InfineonSpice reference

Select a plot and click **Properties**. A window opens and the **Trace properties** are shown. These include parameter in the plot as well as the value.

Attributes

Any changes to color and mark can be reset to default in this menu.

Cut Trace

Select a trace amongst the choice of traces and click **Cut Trace** to delete the trace from the plot.

Copy Trace

Select the trace to copy. Click **Copy Trace**. The trace is on the clipboard. To paste the trace, select the location where you want the trace then, **Trace > Paste Trace**

Delete Trace

Select the trace you want to delete and click **Delete Trace**.

Delete All Traces

Place your cursor in the at once by clicking **Delete All Traces..** A window pops up that asks you if you want to **Delete all traces in the selected plot?YesNo**

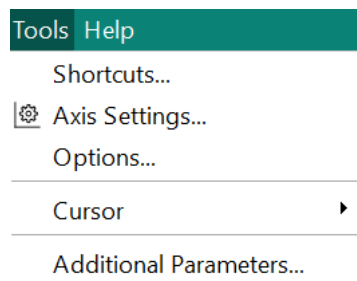
Undelete

You can retrieve a plot or plots by clicking **Undelete** and all traces appear as before delete.

Aliases

You can create an alternative name or an alias for a parameter. Click **Aliases**. A window opens with the name of the parameter. Click in the column next to the parameter and you can enter the alias.

4.4.5 Tools



Shortcuts

Click **Shortcuts** and **Keyboard Shortcuts** opens.

You can filter the large amount of data by **Context** or **Description** or per existing **Shortcut**. If you know what you are looking for, enter the text in the Filter field above the columns. The respective columns below the radio buttons contain:

- **Context** indicates the topic where the shortcut is located
- **Description** explains how to execute the shortcut
- **Shortcut** for the desired outcome

You can enter new shortcuts in a field below the columns. Select the function you wish to edit. Enter your combination for example, alt, ctrl and shift keys and a symbol of your choice or whatever else you like and select **Assign**. The new shortcut for that function will be what you entered.

Edit the function with **Clear**. Select **Default** for the system assigned shortcut.

To reset all the edits you made, click **Restore Defaults** your edits will be replaced with the system assigned shortcuts.

To complete this field, click **OK** to keep your changes or **Cancel** if you decide not to make any changes.

Axis Settings

4 InfineonSpice reference

You can set up your x-axis and y-axis by adjusting the range as well as the steps. Labels for the x-axis and y-axis can be custom assigned or standard assigned. You can also have choose to have a grid with dots or lines in the background. Click **Apply** to see the outcome before you click **OK** to set the changes.

Options

In **Options** you can control

- scrollbars
- colour of your traces
- how the coordinates appear on the trace **Trace Mark Scheme**
- synchronize x-axes
- show the grid in the background
- size of the plot
- Font
- default behavior when closing the workspace - whether to save or not or ask before saving, are options on closing the programme
- You can check the box **Display output variable name and alias**

Cursor

You can apply cursor settings to facilitate navigation and analysis. You can **Display Pointer** as a circle as well as **Display Cursor** which creates dotted lines so that various points on various plots can be seen easier. **Display Cursor** allows a small window to pop up and gives the coordinates of where your cursor is on the plot. As you move your cursor, the ruler moves and shows exact coordinates.

Additional Parameters

The image shows a list of all relevant parameters for the current plot.

4 InfineonSpice reference

Additional Parameters	
Parameter	Value
Date/Time Run	16.04.2025 10:33
Simulator	TITAN
Analysis	tran
Number of X-values	6713
SOURCE	TITAN
VERSION	2025.02.i02
TITLE	GENERATED TITAN NETLIST USING REVISION B2D10AC678
SUBTITLE	NOMINAL
TIME	10:33:01
DATE	2025-04-16
TEMPERATURE	27.0000
ANALYSIS	TRANSIENT ANALYSIS
SERIALNO	12345
ALLVALUES	NO
COMPLEXVALUES	NO
NODES	20
SWEEPVAR	TIME
SWEEPMODE	VAR_STEP
XBEGIN	0.0000000E+00
XEND	1.0000000E-06
FORMAT	1 VOLTSorAMPS;EFLOAT

Close

4.4.6 Help

The **Help** menu contains information on the application and links to helpful resources in the Internet.

User Guide (Waveform Viewer)

Click on **User Guide (Waveform Viewer)** and the instructions for [Evaluating simulations](#) opens.

Ask the Community

Opens a browser window showing the InfineonSpice discussion forums on the Infineon website.

Check for Updates

Opens the InfineonSpice download page on the Infineon website.

If there are newer releases than the one you are using, you can download them from here.

About

Click on **About** and information about your Version in use as well as license information is displayed. Close the window to get back your waveform view.

5 Glossary

Here you find explanations of specific terms that InfineonSpice uses as well as general terms that have specific meanings.

A *component* is a combination of a symbol and a model. It represents a physical component that is used in a schematic to perform a simulation.

A *library* is a data structure that contains one or more models, each of which describes the simulation logic for a component.

A *model* is the simulation logic of a component. It consists of SPICE code and is contained in a library.

A *project* in InfineonSpice is a collection of all elements, such as schematics, components, and simulation settings, that are required to perform a simulation of a particular device. The elements can either be contained in the project or be external and included by reference.

A *schematic* is a data structure in InfineonSpice that represents an electric circuit. It can contain components and their connections.

A *symbol* is the graphical representation of a component in a schematic.

A *view* is a region within the main program window that displays a thematic subset of the program data, such as the workspace, available components, or the simulation environment.

6 Support and feedback

Support for InfiniteonSpice is available on the [Infineon support page](#).



InfiniteonSpice Simulation Tool

An open forum for designers and tool users to ask questions, collaborate and share knowledge. It helps users to connect an...

Additional information about the simulation engine is available in the TITAN documentation. (Coming soon.)

Trademarks

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

Edition 2026-08-07

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-zmz1716901497503**

Important notice

Products which may also include samples and may be comprised of hardware or software or both ("Product(s)") are sold or provided and delivered by Infineon Technologies AG and its affiliates ("Infineon") subject to the terms and conditions of the frame supply contract or other written agreement(s) executed by a customer and Infineon or, in the absence of the foregoing, the applicable Sales Conditions of Infineon. General terms and conditions of a customer or deviations from applicable Sales Conditions of Infineon shall only be binding for Infineon if and to the extent Infineon has given its express written consent.

For the avoidance of doubt, Infineon disclaims all warranties of non-infringement of third-party rights and implied warranties such as warranties of fitness for a specific use/purpose or merchantability. Infineon shall not be responsible for any information with respect to samples, the application or customer's specific use of any Product or for any examples or typical values given in this document.

If this document is marked as 'Preliminary', 'Target', 'Draft' or 'Proposal', Infineon reserves the right to change all information given in this document at any time without notice. Subject to the development and release of a Product for series supply by Infineon, the technical specifications of the Product are set forth in the relevant datasheet provided by Infineon without such marking.

The data contained in this document is exclusively intended for technically qualified and skilled customer representatives. It is the responsibility of the customer to evaluate the suitability of the Product for the intended application and the customer's specific use and to verify all relevant technical data contained in this document in the intended application and the customer's specific use. The customer is responsible for properly designing, programming, and testing the functionality and safety of the intended application, as well as complying with any legal requirements related to its use.

Unless otherwise explicitly approved by Infineon, Products may not be used in any application where a failure of the Products or any consequences of the use thereof can reasonably be expected to result in personal injury.

However, the foregoing shall not prevent the customer from using any Product in such fields of use that Infineon has explicitly designed and sold it for, provided that the overall responsibility for the application lies with the customer. Infineon expressly reserves the right to use its content for commercial text and data mining (TDM) according to applicable laws, e.g. Section 44b of the German Copyright Act (UrhG). If the Product includes security features:

Because no computing device can be absolutely secure, and despite security measures implemented in the Product, Infineon does not guarantee that the Product will be free from intrusion, data theft or loss, or other breaches ("Security Breaches"), and Infineon shall have no liability arising out of any Security Breaches.

If this document includes or references software:

The software is owned by Infineon under the intellectual property laws and treaties of the United States, Germany, and other countries worldwide. All rights reserved. Therefore, you may use the software only as provided in the software license agreement accompanying the software.

If no software license agreement applies, Infineon hereby grants you a personal, non-exclusive, non-transferable license (without the right to sublicense) under its intellectual property rights in the software (a) for software provided in source code form, to modify and reproduce the software solely for use with Infineon hardware products, only internally within your organization, and (b) to distribute the software in binary code form externally to end users, solely for use on Infineon hardware products. Any other use, reproduction, modification, translation, or compilation of the software is prohibited.

For further information on the Product, technology, delivery terms and conditions, and prices, please contact your nearest Infineon office or visit www.infineon.com.