Simulation Tools

How to use ROHM Solution Simulator

Table of Contents

No. 67UG093E Rev.001
February 2025

1. About ROHM Solution Simulator

The ROHM Solution Simulator is an electronic circuit simulation tool that runs on the official ROHM website. It can perform a wide range of simulations on the website from the initial stage of development, including the selection of components and verification of individual devices, to the verification stage on the system level. The simulator allows quick and easy batch verifications of ROHM products, including power device products such as SiC devices as well as IC products such as drive and power supply ICs, with a solution circuit close to an actual environment. Therefore, it can help to significantly reduce working hours during the application development.

Furthermore, the simulator has been developed using "PartQuest™ Explore", which is a simulation platform by Siemens EDA with a proven record in the automobile and industrial equipment industries. As a result, users who have a Siemens account can incorporate simulation data executed on the ROHM Solution Simulator into their own PartQuest™ Explore workspace to perform verifications in a more realistic environment.

This simulator can be used for free and only requires user registration on the official ROHM website. Also, the special site for the ROHM Solution Simulator below provides the documents necessary to access and use the simulator.

The ROHM Solution Simulator is a free web-based simulator that allows batch verifications of power semiconductor devices, ICs, and passive components. It is unique in the industry, featuring the following abilities that can reduce the working hours of electronic circuit and system designers during the application development.

1. Batch verifications of power semiconductors and ICs can be performed with a solution circuit closer to an application environment.

The ROHM Solution Simulator allows precise and easy verifications of ROHM products, including power semiconductors such as SiC devices and IGBTs, ICs such as drive and power supply ICs, and passive components such as shunt resistors, with a solution circuit close to an actual application environment. Characteristics that cannot be observed with individual devices can be simulated collectively along with peripheral circuits.

2. Simulation data can be transferred to a user's own development environment.

Existing users of PartQuest™ Explore and users who newly register an account can incorporate simulation data executed on the ROHM Solution Simulator into their own PartQuest™ Explore environment (workspace). This allows verifications with a system circuit closer to the actual use, including customizations.

PartQuest™ Explore, provided by Siemens EDA

PartQuest™, is a cloud-based software service (SaaS) that can provide various details for building an integrated board design library environment. PartQuest™ Explore, which is part of the SaaS environment, allows you to perform the latest multi-domain simulations and enables access from a web browser. Therefore, you can perform analyses from your PC environment anywhere and anytime in the same manner as at your office. In addition, since the encrypted VHDL/AMS and SPICE models are supported, the data can be imported to the PCB design environment of Siemens EDA (HyperLynx™ AMS, previously Xpedition™ AMS).

2. Operating Environment

To use the ROHM Solution Simulator, the environment must satisfy the following requirements.

Internet access

- Internet access is necessary. The simulator cannot be used offline.
- Depending on the Internet environment of your company, it may not be possible to use the tool because of security countermeasures. (For example, port 80 and 443 are rejected with a firewall.) Contact your IT administrator.

Web browser

- Microsoft Edge, Google Chrome, Mozilla Firefox, Safari, etc.
- If the simulator cannot be loaded, allow cookies in the browser settings.

Monitor resolution

- 1,920 × 1,080 or higher is recommended.

Software installation

- Not necessary

Simulation speed

- Since the simulation engine uses a Siemens cloud server, the simulation speed does not depend on the performance of the local PC.
- The speed may be reduced depending on the usage rate of the cloud server.

3. Restrictions

- Circuit diagrams cannot be edited.*1

- Component properties cannot be changed, with some exceptions.*1

- Circuit diagrams cannot be saved.*1

- Simulation results cannot be saved.*1

- The execution time of simulations is up to 1 hour each (not the TSTOP setting time of the transient analysis).

- Circuit diagrams cannot be downloaded.*1

- The session is ended if no operation is performed for 15 minutes.

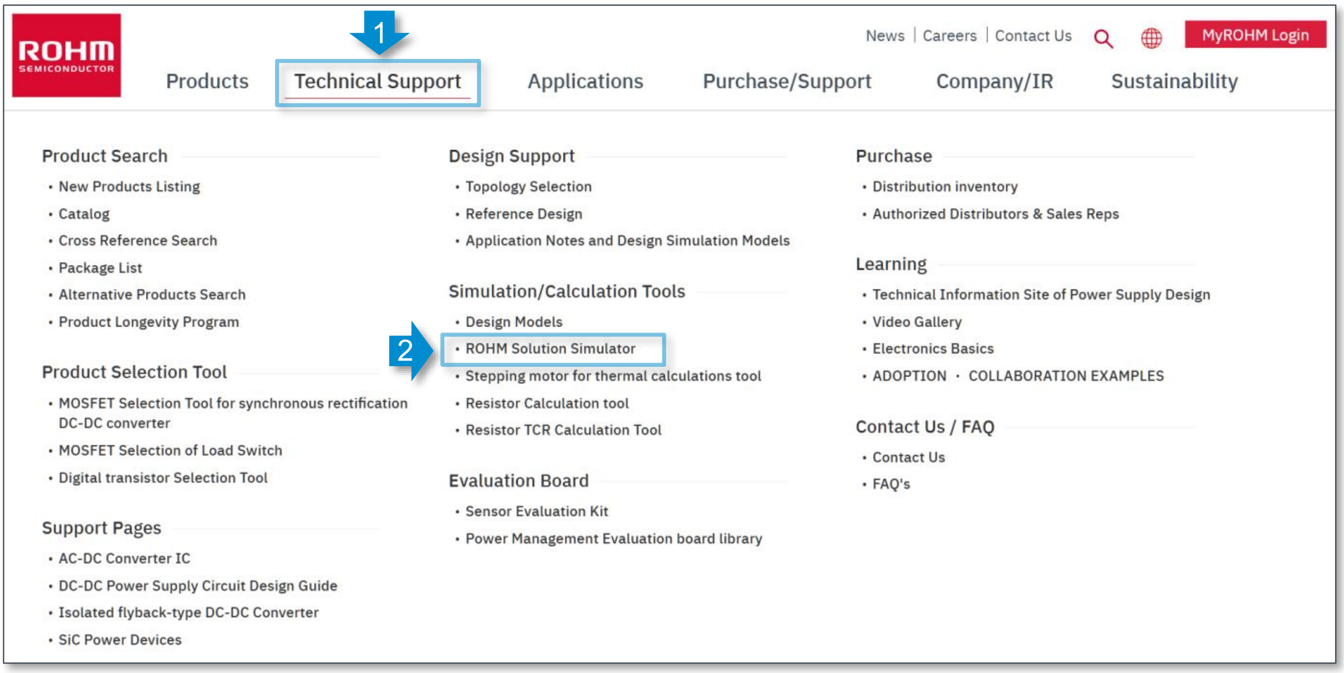
*1: These restrictions are removed by exporting the simulation data to PartQuest™ Explore. See the following sections.

- 12. Exporting Circuit Data > 12-2.
- 12. Exporting Circuit Data > 12-3.

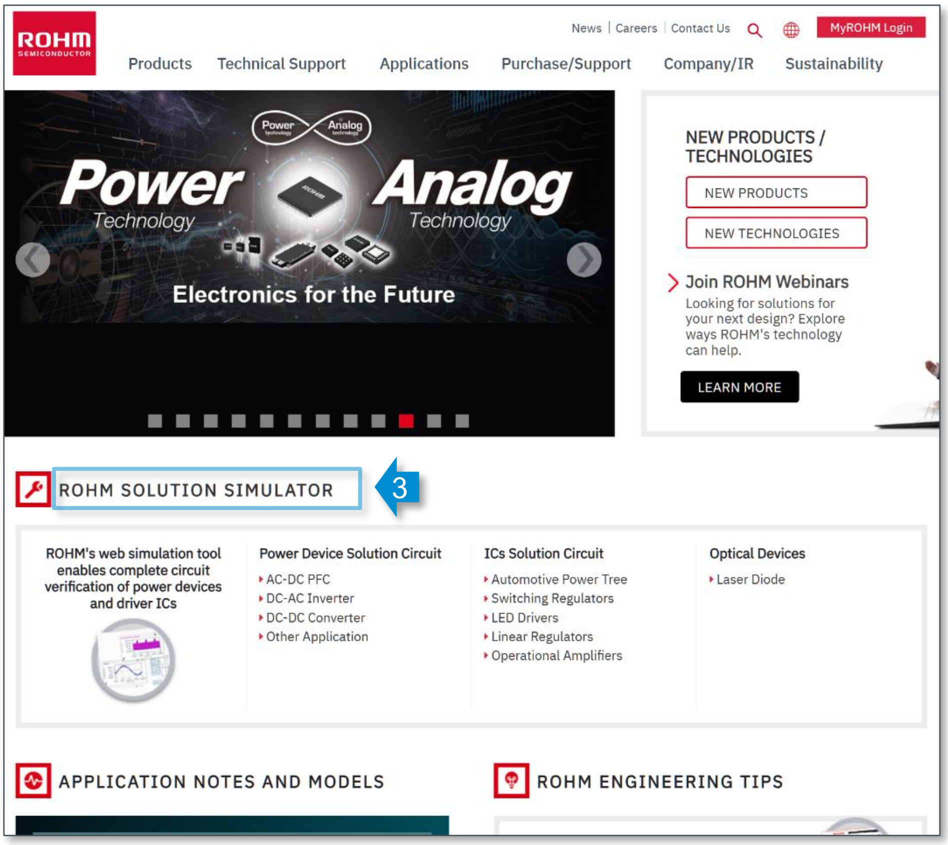
4. Selecting a Simulation Circuit

1. Access the official ROHM website.

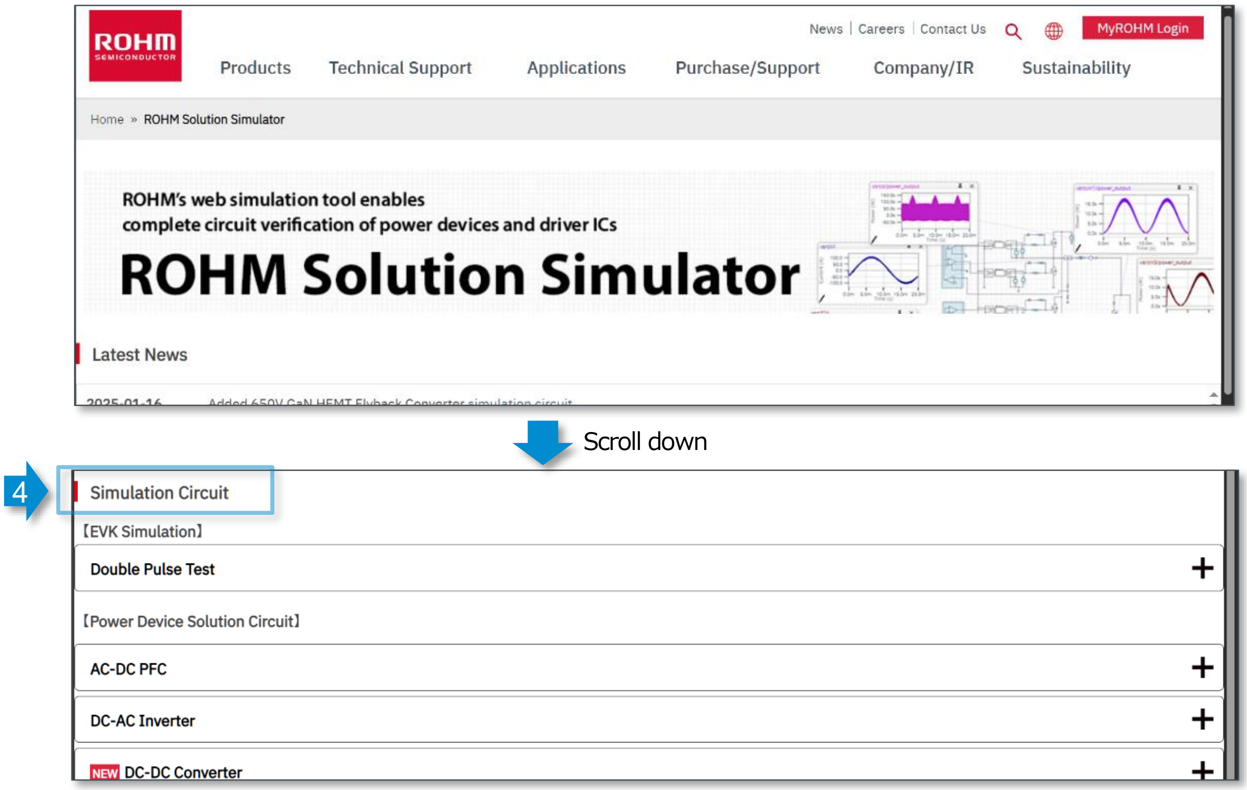
2. Click “Technical Support” (1) in the top menu to expand the list, and then click “ROHM Solution Simulator” (2).



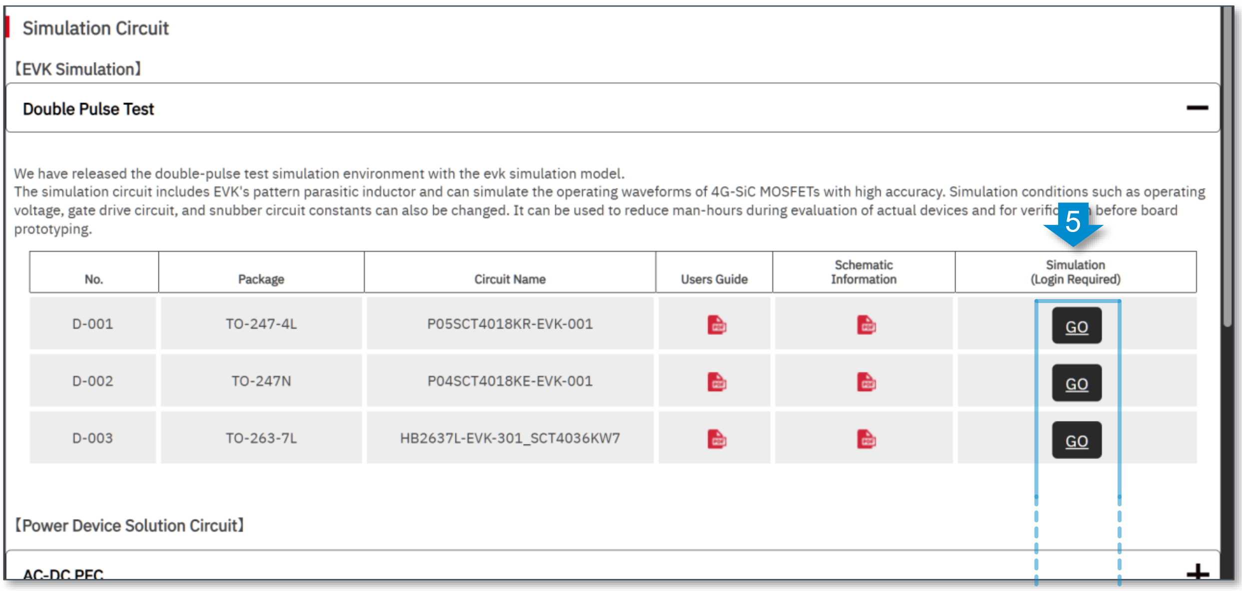
Alternatively, click the special entrance (3) on the top page (as of February 2025).



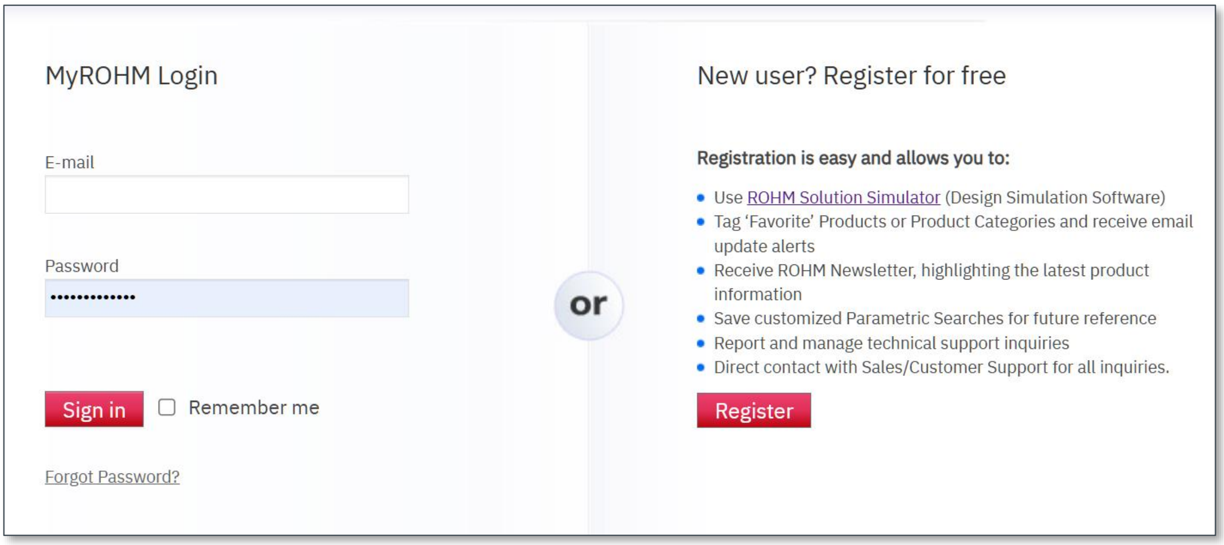
3. When the ROHM Solution Simulator page is displayed, scroll down to find the “Simulation Circuit” section (4).



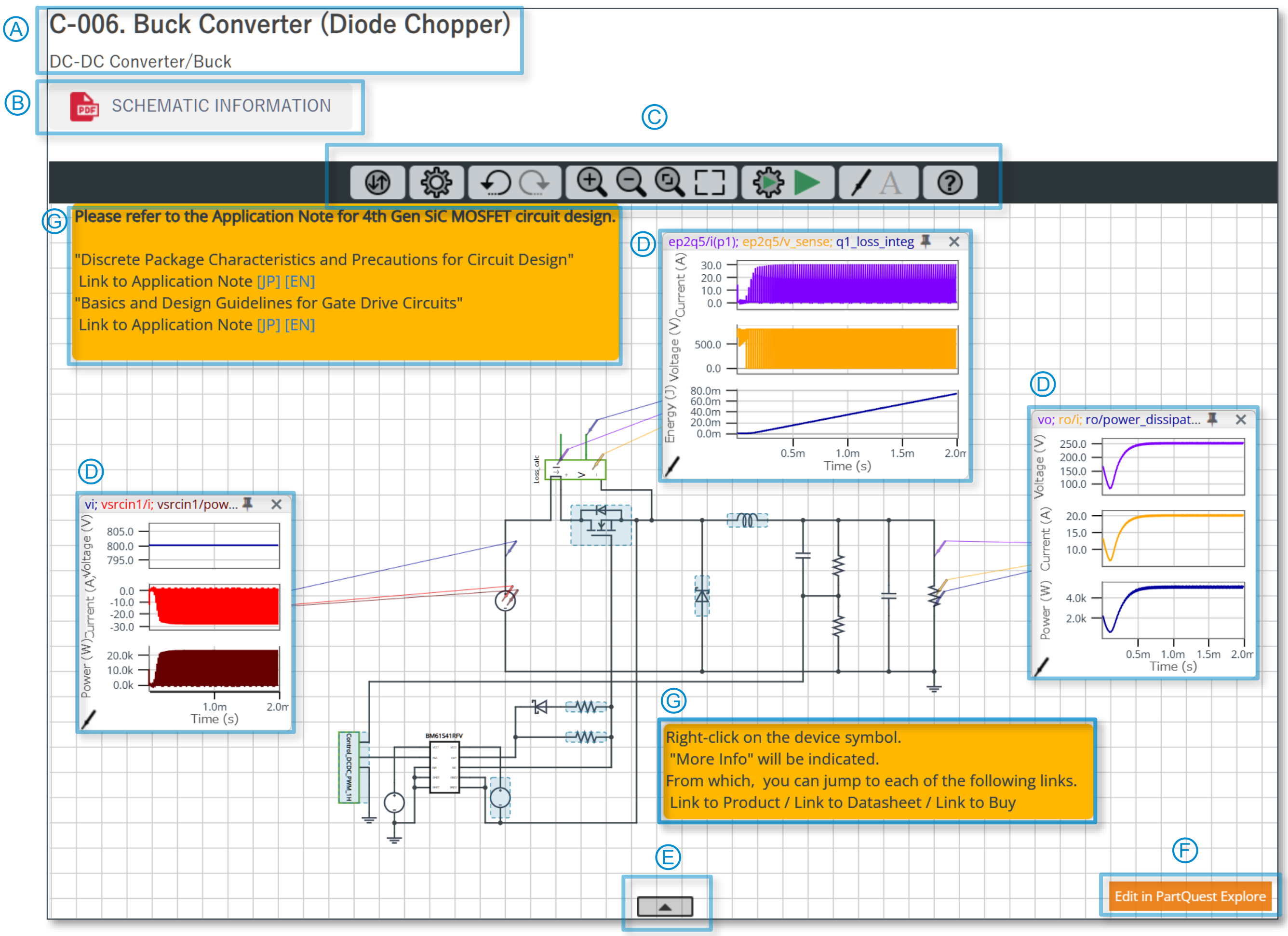
4. Select a circuit to simulate and click “GO” (5).



5. The ROHM Solution Simulator is started (loaded). Log in to MyROHM on the login screen displayed here if you have not logged in. If you do not have a MyROHM account, register for one.



5. Elements of the Main Window



Ⓐ Overview of circuit

Displays the circuit name, topology, overview, etc.

Ⓑ Document of circuit information

Displays the user’s guide for this circuit.

Ⓒ Tool bar

Displays icons for the functions.

Ⓓ Wavebox display

Direct probing to the circuit diagram displays the waveforms inside a small window (box).

Ⓔ Waveform Viewer display button

This viewer enables more advanced waveform display and measurement than the Wavebox.

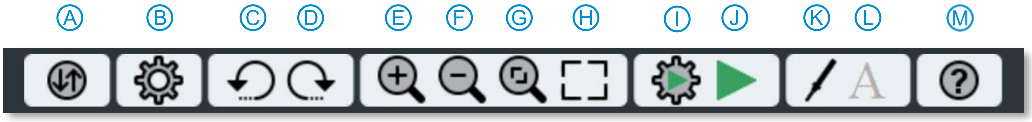
Ⓕ PartQuest Explore export button

Allows you to export circuits from the ROHM Solution Simulator to PartQuest Explore to edit and save circuit diagrams.

Ⓖ Additional information

Displays additional information, if available.

6. Toolbar



Ⓐ Export

Allows you to export the circuit diagram and waveforms and download them as an image.

Ⓑ Design Settings

Allows you to set the global parameters and switch between displaying/hiding the component labels.

Ⓒ Undo

Cancels the previous operation.

Ⓓ Redo

Performs the canceled operation again.

Ⓔ Zoom In

Magnifies the display.

Ⓕ Zoom Out

Compresses the display.

Ⓖ Zoom to Fit

Fits the entire schematic into the display area.

Ⓗ Enter Fullscreen Mode

Starts the full screen display.

Ⓘ Simulation Settings

Allows you to configure various simulation settings.

Ⓙ Run Simulation

Performs a simulation.

Ⓚ Waveform Probe

Dragging and dropping the probe icon in the circuit diagram displays the waveforms.

Ⓛ Add annotation

This function allows you to add annotations with texts and graphics. However, it is not available in the ROHM Solution Simulator.

Ⓜ Help

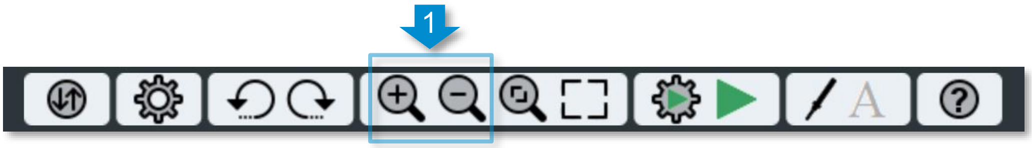
Displays the help function (available in English only).

7. Schematic Display Operation

This chapter explains the basic operation method of the circuit diagram display.

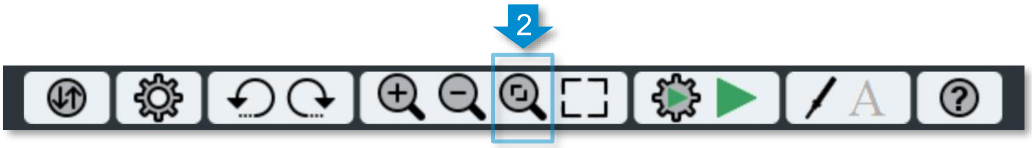
1. Zoom in and out

You can zoom in or out the diagram using the tool bar or mouse wheel. Every time you click the magnifying glass icon for "Zoom In" or "Zoom Out" (1) on the tool bar, the diagram is zoomed in or out with reference to the center of the screen (circuit diagram display area). Alternatively, rotating the mouse wheel zooms in or out the diagram with reference to the mouse pointer position.



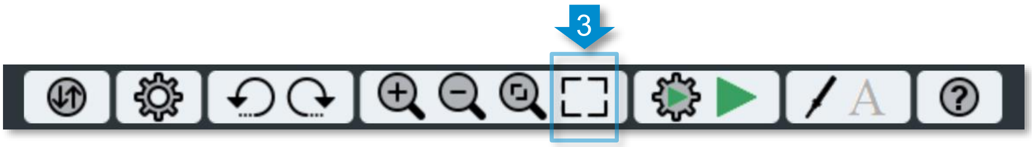
2. Entire display

Clicking the magnifying glass icon for "Zoom to Fit" (2) on the tool bar zooms the circuit diagram so that the entire diagram fits the screen.

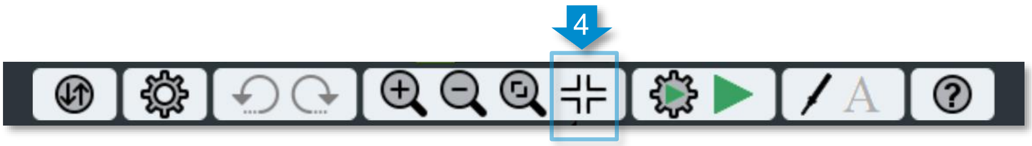


3. Full screen mode

Clicking the "Enter Fullscreen Mode" icon (3) on the tool bar starts the full screen display, maximizing the work area.

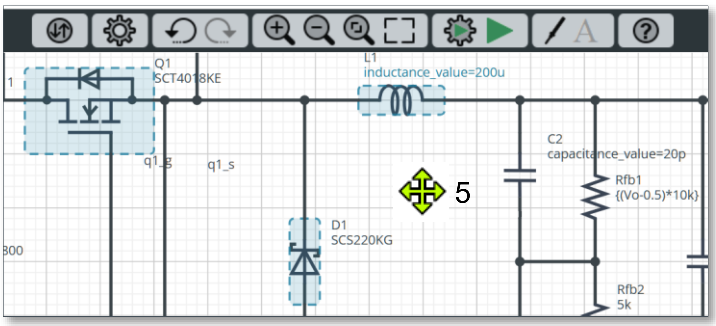


To end the full screen display, press the "Esc" key or click the "Exit Fullscreen Mode" icon (4) on the tool bar.



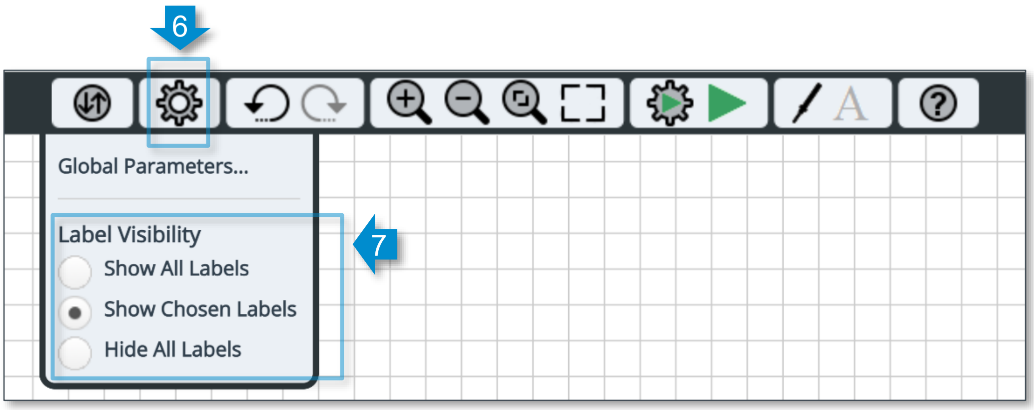
4. Moving the display position

Pressing the left mouse button at a desired position in the circuit diagram changes the pointer from the arrow to the "move" cursor (5). Moving the mouse while holding down the left button moves the display position.

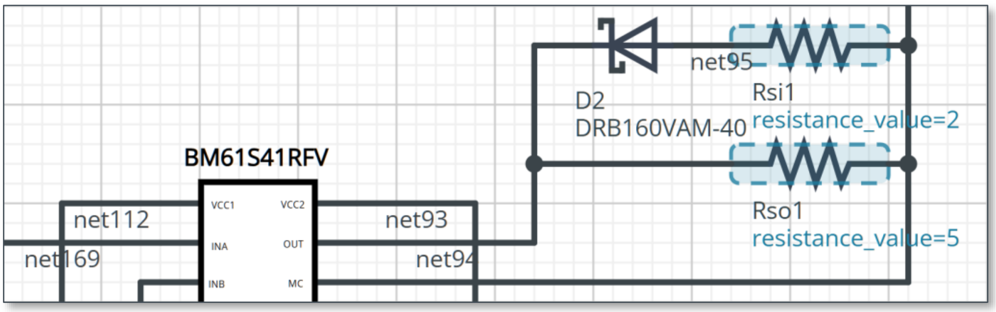


5. Label display

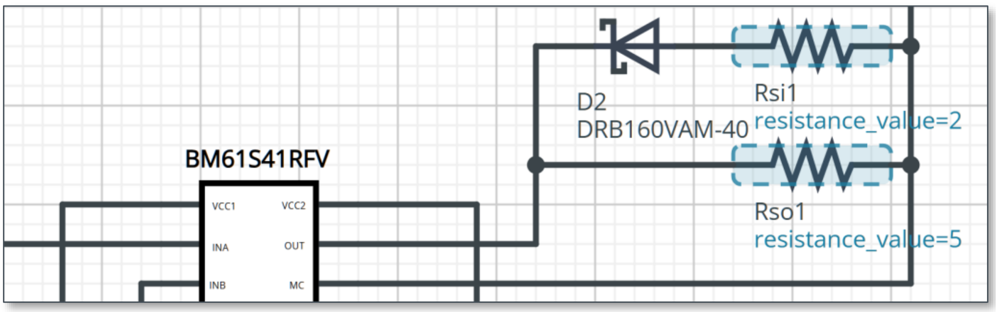
Click the "Design Settings" icon (6) on the tool bar and then select an option from "Label Visibility" (7).



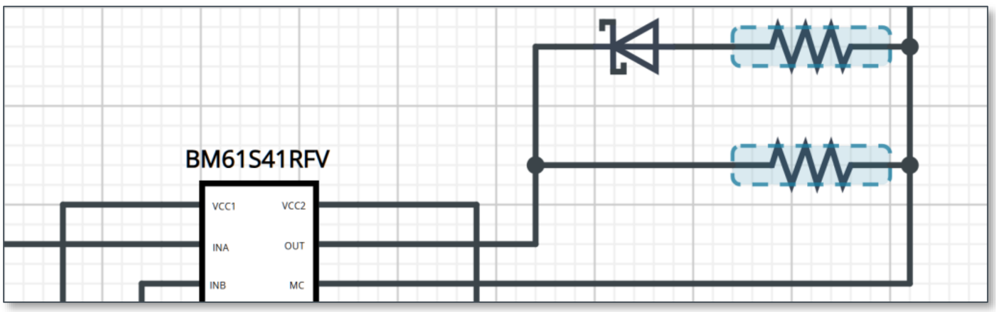
- Show All Labels: Displays the labels of all components and nets.



- Show Chosen Labels: Displays the labels of all component instance names, all component instance properties that have been set to be shown, and all user-defined net names.

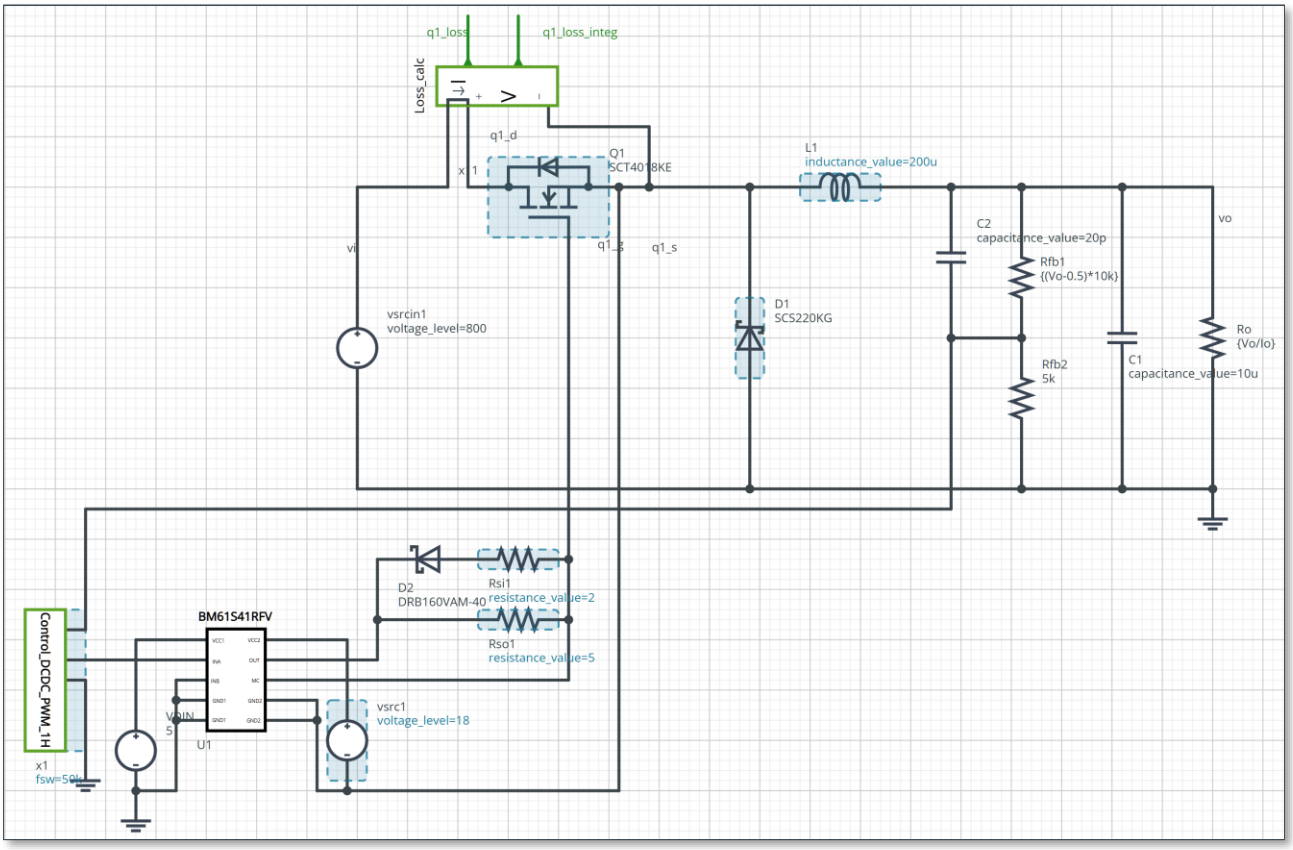


- Hide All Labels: Hides the labels of all components and nets.

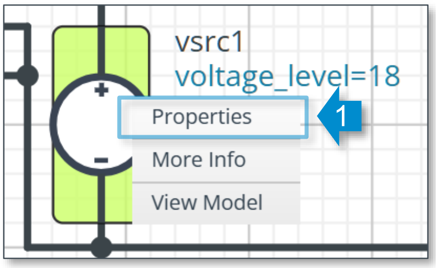


8. Change Component Properties

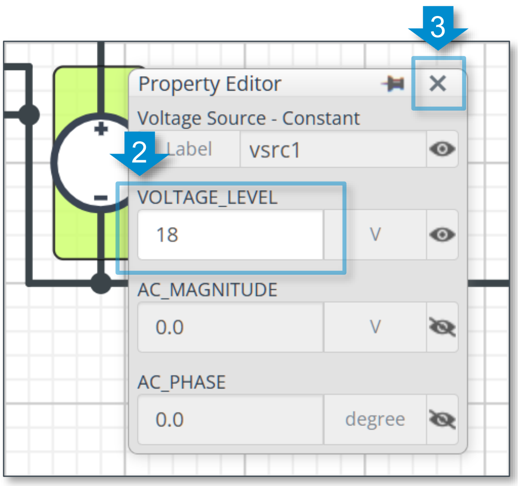
In principle, you cannot edit circuit diagrams with the ROHM Solution Simulator. However, it is possible to change only the component properties enclosed in blue.



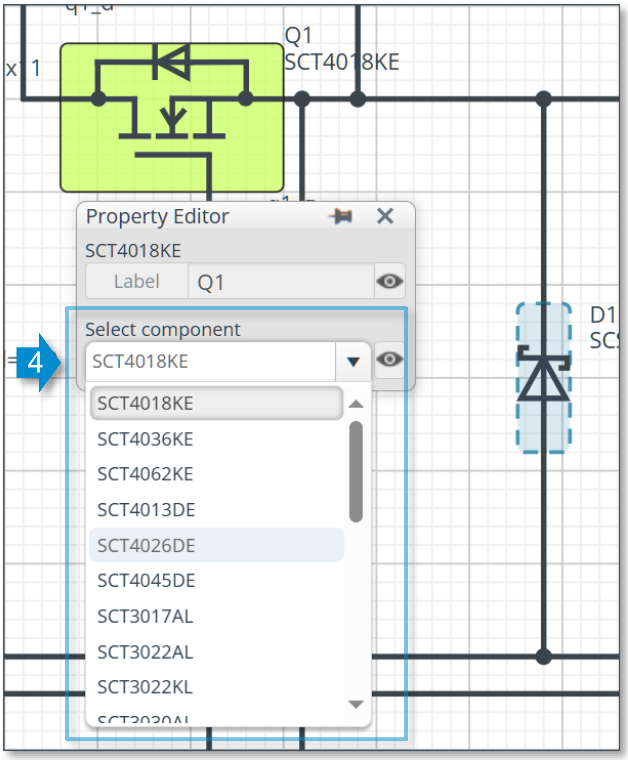
1. Double-click a component whose properties you wish to change. Alternatively, right-click on the component and then select "Properties" (1) in the context menu. The selected component turns green.



2. "Property Editor" is opened. The properties with a white background can be changed (2). A pop-up message indicates the range of the numerical value that can be input. Although a numerical value can be directly input in the example shown on the right, it may be selected from a drop-down list in other cases. After completing the input, click "×" (3) on the upper right to close "Property Editor".

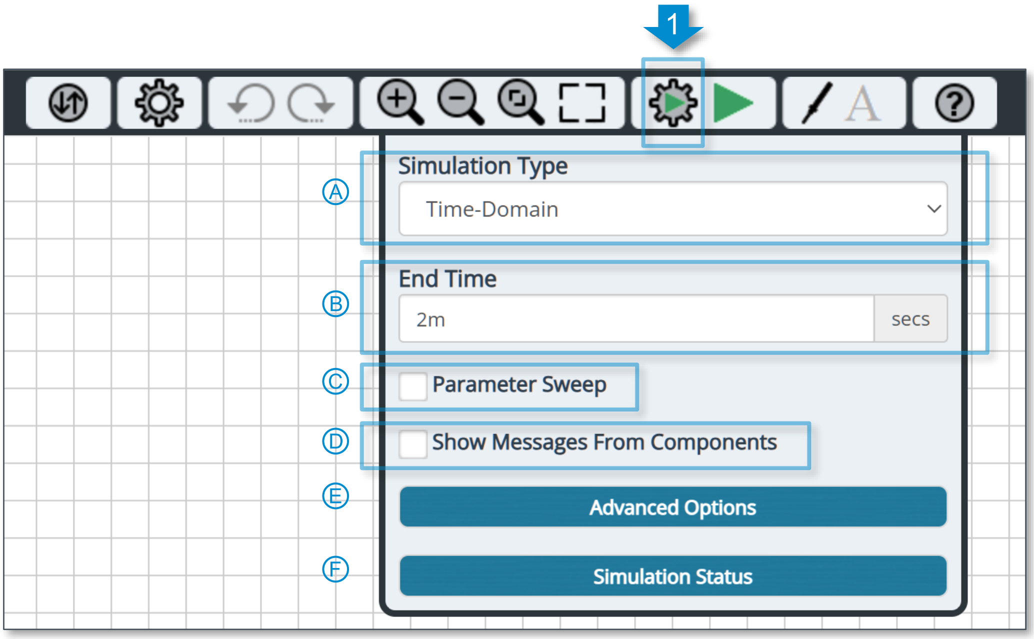


3. For power devices such as transistors and diodes, a product can be selected from the drop-down list (4). This allows you to evaluate differences in the device characteristics in the same simulation environment.



9. Simulation Settings

The simulation settings have been preset according to the circuits and it is normally not necessary to change the settings. However, to perform a simulation with different settings, click the "Simulation Settings" icon (1) on the tool bar to display the drop-down dialog.



Ⓐ Simulation Type

Since this parameter has been set according to the circuit, you cannot obtain any simulation results if you change it. One of the following has been set.

- Time-Domain: transient analysis
- Frequency-Domain: frequency analysis
- DC: DC analysis

Ⓑ Settings specific to the simulation types

The following settings can be configured for each simulation type.

- Time-Domain: set "End Time".
- Frequency-Domain: set "Start Frequency" and "End Frequency".

Although SI suffixes (u, m, k, MEG, G, etc.) can be added, do not add units. The unit is displayed on the right of the time/frequency input field. The setting that is specified in "Manual Options" has priority.

Ⓒ Parameter Sweep

Sweeps the parameters with the properties selected with the component property editor.

Ⓓ Show Messages From Components

Messages created from the components are displayed in a table.

Ⓔ Advanced Options

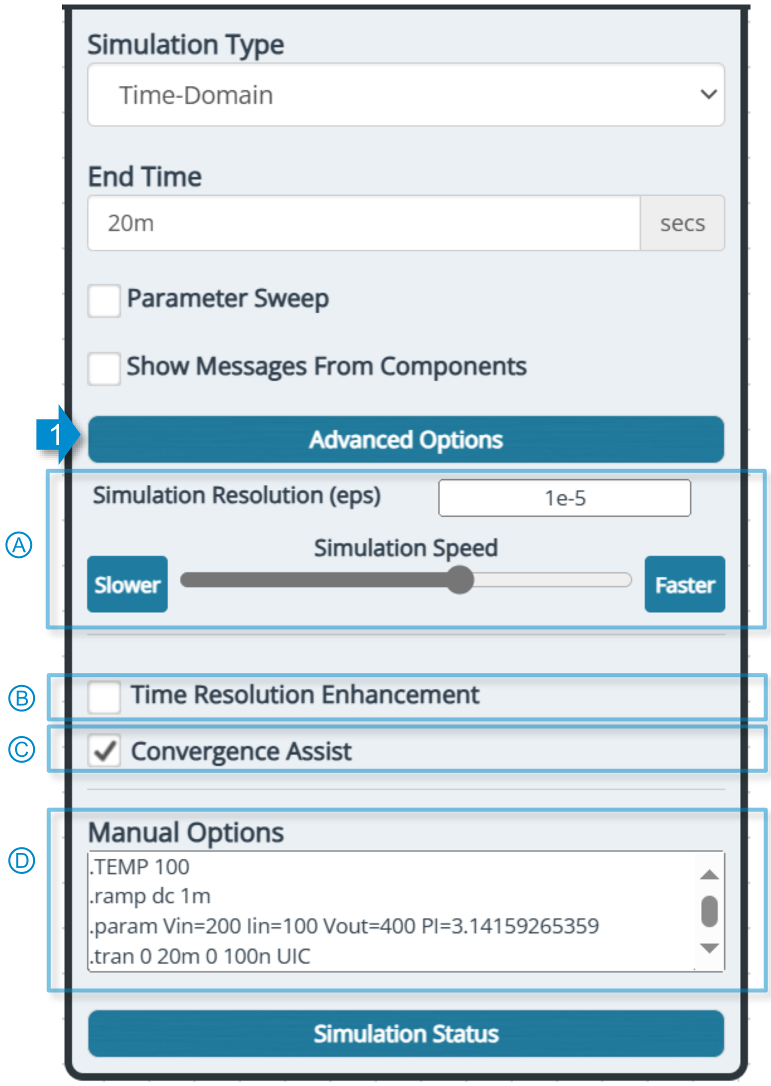
Allows you to set other advanced options.

Ⓕ Simulation Status

Displays the simulation status and results. After the simulation is completed, the net list can be downloaded.

9-1. Advanced Options

Click "Advanced Options" (1) to expand the drop-down dialog.



Ⓐ Setting the simulation resolution and speed

Since the optimum values have been set, it is usually not necessary to change the values. However, they can be changed to solve detailed calculations and convergence problems. Move the slider to adjust the epsilon (eps) value. The larger the epsilon value, the faster the simulation. However, the precision is decreased.

Ⓑ Time Resolution Enhancement

Transient phenomena at high speed and high frequencies can be followed more accurately by decreasing the time step.

Ⓒ Convergence Assist

Allows you to change the simulation algorithms so that the solution of the system can be converged.

Ⓓ Manual Option

This is an option to perform simulations with descriptions. The settings can be configured in more detail. This option is recommended for expert users.

9-2. Manual Options

This section explains commands (instructions) that can be described with the manual option. These commands are not case-sensitive.

Initial Condition

The following initial condition options are available for the ROHM Solution Simulator.

.IC Command

The .ic command specifies the node voltages to be used for performing the DC (operating point) analysis. After the DC (operating point) analysis, the voltage specified with the .ic command is used as the start value for the transient analysis.

Examples: Specifies that for the duration of the DC analysis, the voltages at the node v_out be fixed to 5V.

```
.ic v(v_out) = 5
```

Note that the representation of “`v()`” is optional. The above can also be described as “`.ic v_out = 5`”.

If there are several settings, you can describe them in separate lines or in the same line using spaces as delimiters.

```
.ic v(v_in1) = 2
.ic v(v_out3) = 4      .ic v(v_in1) = 2 v(v_out3) = 4 v(net56) = 7
.ic v(net56) = 7
```

.NODESET Command

The .nodeset command specifies the node voltages to be used for calculating DC solutions. The specified values are used to calculate the initial DC solution and then the second and final DC solution is calculated without using the .nodeset command.

This command is helpful if an approximate DC solution of the circuit is known. It allows the simulator to converge to the exact solution. For a circuit with multiple operating points, the .nodeset command may be necessary to obtain convergence to a certain solution.

Example: According to the examples in “.ic Command”.

.GUESS Command

Although the .guess command is similar to the .nodeset command, the specified node voltages are maintained during the initial iterative calculation only. Subsequently, the node voltages are calculated by the simulator.

Examples: According to the examples in “.ic Command”.

.OPTION Command

Options are available for changing certain methods in case the default methods fail to obtain convergence and find the DC operating point of the circuit. The table of optional settings is shown below.

Examples: Set the numerical integration method to the Gear method.

```
.option method = gear
```

If there are several settings, you can describe them in separate lines or in the same line using spaces as delimiters.

```
.option method = gear
.option hmax = 100n
.option eps = 1e-7
```

```
.option method = gear hmax = 100n eps = 1e-7
```

Option	Description	Syntax

ABSTOL	Absolute current accuracy.	`.option abstol = <value>`
		
		Arguments
		`<value>`
		
		Value of the absolute current accuracy.
		Default is VNTOL × ITOL.

BE	Forces Backward Euler integration to be used. Trapezoidal integration is used by default.	`.option be` Arguments None.

CHGTOL	Specifies the absolute tolerance on charge and is used by charge control devices. This option is only used by the “Gear” algorithm or if option QTRUNC is set.	`.option chgtol = <value>`
		
		Arguments
		`<value>`
		
		Specifies the absolute tolerance on charge.
		Default value is set by EPS.

DEFPTNOM	Enables a parameter to be defined with the name TNOM. In such a case, this value will be used inside parameter expressions instead of the default TNOM or the value set using option TNOM. This option must be specified at the top of the netlist.	`.option defptnom` Arguments None.

Option	Description	Syntax

EPS	Sets the internal simulator accuracy.	`.option eps = <value>`
		
		Arguments
		`<value>`
		
		Internal simulator accuracy.
		Default value is 1 mV or 5 mV depending on the voltage source levels. Values smaller than 1.0e−10 V need to be defined with the .OPTION UNBOUND parameter.

FLUXTOL	Specifies the absolute tolerance on flux. This option is only used by the “Gear” algorithm or if option QTRUNC is set.	`.option fluxtol = <value>`
		
		Arguments
		`<value>`
		
		Default value is set by EPS.

GEAR	Forces Gear integration to be used. Trapezoidal integration is used by default. Take care with the gear algorithms as you may potentially miss some oscillations in circuits.	`.option gear` Arguments None.

GMIN	Sets the conductance value that is placed in parallel with all PN junctions and drain and source nodes of MOSFET models.
	Set GMIN to enhance the convergence properties that are degraded by having too low a value of off conductance for PN junctions and MOSFET devices. Large values of GMIN may cause unreasonable circuit response.
		`.option gmin = <value>`
		
		Arguments
		`<value>`
		
		Default value is 1e−12.

GRAMP	Ramps the conductance value that is placed in parallel with all PN junctions of all devices, and all drain and source nodes of MOSFET models, from GMIN×10^{GRAMP} down to GMIN. This helps DC convergence in some circuits. This option may also be used in conjunction with the .RAMP command.	`.option gramp = <value>`
		
		Arguments
		`<value>`
		
		Default value is 0.

GSHUNT	Add a conductance of value val between each node and ground. This can be used to eliminate oscillation problems caused by numerical noise or high frequency oscillation.	`.option gshunt = <value>`
		
		Arguments
		`<value>`
		
		Default value is 0.

HMAX	Sets the maximum internal timestep.	`.option hmax = <value>`
		
		Arguments
		`<value>`
		
		Default value is 1/10 of the wave period when using SIN and SFFM functions.

No. 67UG093E Rev.001
February 2025

Option	Description	Syntax

HMIN	Sets the minimal internal timestep.	`.option hmin = <value>`
		Arguments
		`<value>`
		Default value is 1 ps, which is a value well suited for typical MOS circuits.

ITL1	Sets a limit on the maximum number of DC iterations.	`.option itl1 = <value>`
		Arguments
		`<value>`
		Number of iterations. Default value is 100.

ITL3	Used for timestep control. If convergence is reached in less than the number of iterations specified, the next timestep is doubled.	`.option itl3 = <value>`
		Arguments
		`<value>`
		Number of iterations. Default value is 3.

ITL4	Used for timestep control. If convergence is not reached within the number of iterations specified, the present timestep is rejected and the next one reduced by FT.	`.option itl4 = <value>`
		Arguments
		`<value>`
		Number of iterations. Default value is 13.

ITL6	Used for timestep control. If convergence is reached in less than the number of iterations specified, the next timestep is doubled. Equivalent to ITL3 when DC convergence assist is in use.	`.option itl6 = <value>`
		Arguments
		`<value>`
		Number of iterations. Default value is 5; however this default is 6 in the Pseudo-Tran algorithm.

ITL7	Used for timestep control. If convergence is not reached within the number of iterations specified, the present timestep is rejected and the next one reduced by FT. Equivalent to ITL4 when DC convergence assist is in use.	`.option itl7 = <value>`
		Arguments
		`<value>`
		Number of iterations specified. Default value is 30; however this default is 20 in the Pseudo Tran algorithm.

ITL8	Used for timestep control. Controls the maximum number of iterations allowed for each trial of ramping algorithm (GRAMP, PSTRAN, .RAMP DC).	`.option itl8 = <value>`
		Arguments
		`<value>`
		Number of iterations. Default value is 10000.

Option	Description	Syntax

ITOL	Controls the current accuracy of the simulator when solving circuits using Newton iterations.	`.option itol = <value>`
		
		Arguments
		`<value>`
		
		Default value is 1e−6 A/V.

MAXORD	Used in conjunction with option GEAR. Specifies the maximum order of the Gear integration method and must be in the range 1 to 6. The value 1 is equivalent to the BE option.	`.option maxord = <value>`
		
		Arguments
		`<value>`
		
		Default value is 2.

METHOD	Specify the numerical integration method.	`.option method = <command>`
		
		Arguments
		`<command>`
		
		be: Forces "backward Euler" integration to be used.
		gear: Forces "Gear" integration to be used.
		trap: Forces "trapezoidal" integration to be used.
		
		Trapezoidal integration is used by default.

PSTRAN	Forces the simulator to use the PSTRAN (PSeudo TRANsient) algorithm prior to any other convergence aid. This algorithm is one of the DC convergence algorithms that are automatically used by the simulator when the standard DC algorithm fails to converge.	`.option pstran` Arguments None.

RELTOL	Controls both the timestep size and the accuracy of Newton iterations.	`.option reltol = <value>`
		
		Arguments
		`<value>`
		
		Default value is 1e−3.

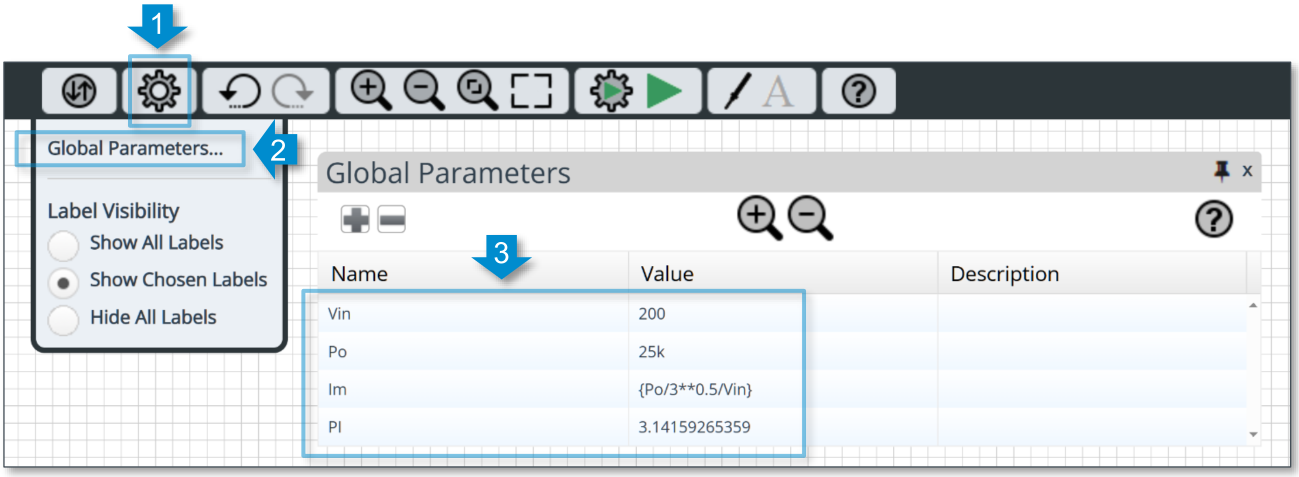
STEP	Imposes a fixed timestep to be used by the simulator as defined by the val value. By default, the simulator uses a varying timestep.	`.option step = <value>`
		
		Arguments
		`<value>`
		
		No default value.

TMAX	Overwrites the default maximum device temperatures.	`.option tmax = <value>`
		
		Arguments
		`<value>`
		
		Maximum device temperatures, specified in degrees Celsius. Default value is 500 °C.

Option	Description	Syntax
TMIN	Overwrites the default minimum device temperatures.	.option tmin = <value> Arguments <value> Minimum device temperatures, specified in degrees Celsius. Default value is −273 °C.
TNOM	Sets the default model reference temperature for parameters provided in the .MODEL statement. TNOM is the temperature at which the given model parameters are valid. If the temperature differs from TNOM, model parameters are internally recalculated to reflect the temperature dependence of the simulated devices.	.option tnom = <value> Arguments <value> Default value is 27 °C.
TRTOL	Used for timestep control. TRTOL is a factor that estimates the amount of error introduced in truncating a series used in the algorithm. This error is a reflection of what the minimum value of the timestep should be to reduce simulation time and maintain accuracy. The larger TRTOL is, the larger the timestep will be.	.option trtol = <value> Arguments <value> Default value is 7.0.
VMIN VMAX	Sets the minimum and maximum voltage values for which the simulator searches for the DC operating point of a circuit. Power supply levels are very often parametrized, and the values of VMIN and VMAX typically depend on the power supply. Therefore VMIN and VMAX can have their values specified by parameters.	.option vmin = <value> | <expression> .option vmax = <value> | <expression> Arguments <value> Voltage value. No default value; computed from the circuit. <expression> Voltage expression. Examples This example shows the voltage values specified as expressions: .param vx = 1.0 .option vmin = '-100 * vx' .option vmax = '100 * vx'
VNTOL	Controls the voltage accuracy of the simulator when solving circuits using Newton Raphson techniques.	.option vntol = <value> Arguments <value> Default value is 1 1 μV.

.PARAM Command

You can create user-defined parameters using the .param command. In addition to using the "Manual Options" field, you can also set the user-defined parameters (3) by clicking the "Design Settings" icon (1) on the tool bar and then selecting "Global Parameters..." (2). Changes made in either setting are applied to both.



Example: Set "400" for parameter "V_IN".

```
.param V_IN = 400
```

If there are several settings, you can describe them in separate lines or in the same line using spaces as delimiters. However, if "Global Parameters" is displayed after the settings are specified in the same line, the description in "Manual Options" is reformatted to separate lines.

```
.param Po = 5k
.param Vo = 200                 .param Po = 5k Vo = 200 Im = {Po/Vo/3**0.5}
.param Im = {Po/Vo/3**0.5}
```

In addition, you cannot use the plus sign to connect several lines into a single command as shown below.

```
.param Po = 5k
+ Vo = 200
+ Im = {Po/Vo/3**0.5}
```

.TEMP Command

The temperature for circuit simulation. The .TEMP command works for all analysis types.

Default value is 27°C (300.15K).

Examples: Specifies circuit analyses at 100°C.

```
.TEMP 100
```

.RAMP Command

Automatic Ramping. This command is used when the simulator encounters difficulties finding a DC operating point with the conventional .DC command. There are two options available:

- DC ramping may be performed if the power supplies in a circuit are increased linearly from time 0, by a fixed voltage step. At each step, a DC operating point is searched up to the maximum power supply voltage.
- Transient ramping may be performed if the power supplies are increased linearly from time 0 to T1, at which point simulation is continued in the form of a transient analysis. When time T2 has been reached, a DC operating point is searched for by the simulator.

Usage:

```
.ramp DC <value>
.ramp TRAN <T1><T2>
.ramp DC 10
.ramp TRAN 1u 2u
```

Parameters:

DC: Keyword indicating that DC ramping should be performed.

TRAN: Keyword indicating that transient ramping should be performed.

<*value*>: Voltage step at which DC ramping is carried out in volts. The ramping process increases the DC source from 0 up to the nominal value. <*value*> is the largest step that can be used. The default value is 0.1V.

<*T1*>: The time at which simulation should be continued. The default value is 1µs.

<*T2*>: The time at which the DC operating point is searched for. The default value is 10s.

Analysis Types

This parameter is usually set with the "Simulation Settings" icon on the tool bar. However, you can also describe the parameter in "Manual Options" if other settings are required. The setting specified in "Manual Options" has priority.

.AC

Usage:

```
.ac <TYPE> <nb> <fstart> <fstop> [UIC]
.ac DEC 10 1 100MEG UIC
```

Parameters:

<*TYPE*>: Can be one of the following.

- DEC: Keyword to select logarithmic variation.
- OCT: Keyword to select octave variation.
- LIN: Keyword to select linear variation.

<*nb*>: Number of points per decade or octave or points over the range from fstart to fstop.

<*fstart*>: Start frequency in hertz.

<*fstop*>: Stop frequency in hertz.

UIC: If UIC is specified, no DC analysis is performed before the AC analysis.

.TRAN

Usage:

```
.tran <TPRINT> <TSTOP> [TSTART[HMAX]] [UIC]
.tran 0 20m 0 100n UIC
```

Parameters:

<*TPRINT*>: Plotting increment for a waveform (in seconds).

<*TSTOP*>: The transient analysis duration (in seconds).

[*TSTART*]: No outputs are stored from 0 to TSTART seconds.

[*HMAX*]: Sets the maximal internal timestep. When HMAX is specified both in the .OPTION command and in the .TRAN command, the HMAX in .OPTION is considered by ROHM Solution Simulator.

UIC: Keyword that indicates that you do not want ROHM Solution Simulator to solve for the quiescent operating point before beginning the transient analysis (Use Initial Conditions). ROHM Solution Simulator automatically initializes all the node voltages itself as well as any user- defined initial node voltages included in a .IC command. The UIC option is recommended for the simulation of astable or very large digital circuits.

Parameter Names

Parameter names can contain an arbitrary sequence of alphanumeric and special characters (%, $, #, _). Parameter names cannot be broken at the end of a line. Parameter names should not contain boolean operators to prevent ambiguity (!=, ==, <, <=, ||, &&).

Reserved Keywords

The .cir and .cmd files reserve certain keywords.
The following keywords could appear in expressions. However, they cannot be included in a .PARAM command.

AMNOISE	BFACTOR	BOPT	FREQ	GA	GAC	GAM
GAMMA_OPT	GASM	GAUM	GOPT	GP	GPC	INOISE
KFACTOR	LSC	MUFACTOR	NFMIN	ONOISE	PHI_OPT	PHNOISE
POWER	RNEQ	SCALE	SNF	SSC	TEMP	TGP
TIME	TNOM *1	XAXIS				

If any .PARAM is named with one of these keywords, it will be rejected as an error. For example, the following statement will generate an error:

```
.param power = 100
```

*1: TNOM could be specified as a parameter in a .PARAM command when .OPTION DEFPTNOM is set.

Numerical Values and Scale Factors

Values are always handled as real numbers. They can be specified in exponential notation or with scale factors (alphabetic multiplier prefixes). For scaling, you can choose between exponential notation, or use the following alphabetic multiplier prefixes:

Numerical value	Exponential notation	Alphabetic Multiplicative prefix
1.0×10^{-18}	1.0e-18	1.0a, 1.0A
1.0×10^{-15}	1.0e-15	1.0f, 1.0F
1.0×10^{-12}	1.0e-12	1.0p, 1.0P
1.0×10^{-9}	1.0e-9	1.0n, 1.0N
1.0×10^{-6}	1.0e-6	1.0u, 1.0U
1.0×10^{-3}	1.0e-3	1.0m, 1.0M
1.0×10^{3}	1.0e3	1.0k, 1.0K
1.0×10^{6}	1.0e6	1.0meg, 1.0MEG
1.0×10^{9}	1.0e9	1.0g, 1.0G
1.0×10^{12}	1.0e12	1.0t, 1.0T

Notes on Alphabetical Scale Factors

Alphabetic characters as scale factors have the following restrictions:

- The meter-kilogram-second (MKS) system of units is used throughout the netlist.
- Alphabetic characters that are not scale factors are ignored if they immediately follow a number. For example, 10, 10V, and 10Hz all represent the same number, 10. However, 10A will be interpreted as 1.0e-17, because A is interpreted as the atto prefix (10^{-18}).
- Letters immediately following a scale factor are ignored. Thus M, MA, MSEC, and MMHOS all represent the same scale factor, M.
- The letter M is the prefix for the scale factor of 1×10^{-3}. The letters MEG are the prefix of the scale factor of 1×10^{6} (mega). This is commonly confused in SPICE syntax (there is no case sensitivity to distinguish "m" and "M").
- Multiplier prefixes are not cumulative. For example, KK is not interpreted as MEG, but still as K (1×10^{3}) the second letter is ignored.

Operators

This section contains information on mathematical operators you can use in expressions.

Operator Precedence

The order of precedence and associativity of operators in ROHM Solution Simulator affect the evaluation of expressions. Expressions with higher-precedence operators are evaluated first. The following table Operator Precedence summarizes the precedence and associativity (the order in which the operands are evaluated) of operators, listing them in order of precedence from highest to lowest. Where several operators appear together, they have equal precedence and are evaluated according to their associativity.

Operator	Description	Associativity
( )	function call	left-to-right
!, -	logical NOT, unary negation	right-to-left
**, ^	power, power (synonym)	left-to-right
*, /	multiply, divide	left-to-right
+, -	add, subtract	left-to-right
<<, >>	bitwise left shift, bitwise right shift	left-to-right
<, <=, >, >=	less than, less than or equal, greater than, greater than or equal	left-to-right
==, !=	equal, not equal	left-to-right
&	bitwise AND	left-to-right
|	bitwise OR	left-to-right
&&	logical AND	left-to-right
||	logical OR	left-to-right

Arithmetic Operators

The arithmetic operators available are +, -, *, / and ^ (or **) for power, the power operator (^) has the highest precedence.

Boolean Operators

The following table lists the available Boolean expressions/operators.

Operator	Meaning
!=	not equal to
==	equal to
<	less than
<=	less than or equal to
>	greater than
>=	greater than or equal to
||	OR operator
&&	AND operator

Bitwise Operators

The following table lists the available bitwise operators.

Operator	Meaning
&	Bitwise AND operator
|	Bitwise OR operator
<<	Bitwise shift left operator
>>	Bitwise shift right operator

Expressions

You can use expressions in a netlist with certain restrictions. Numerical expressions must be contained within braces { }, single quotes ('), or parentheses ( ). However, you should enclose string expressions within quotation marks ("). Mathematical grouping within expressions must be done using normal parentheses ( ). You can use constants and parameters in expressions, together with the built-in functions and operators described in Mathematical Functions. You can use expressions in the following situations:

- Parameters in the calculation of MOS geometries and R, C and L values
- Parameter values in the .MODEL command
- Time point values in the signal descriptions PULSE, PWL, SFFM, SIN and EXP
- Voltage and current source values
- .PARAM, .EXTRACT and .DEFWAVE commands
- E and G sources described by functions or tables
- R, C and L devices described by functions

Expression Examples

```
rfb= 1 2 {(Vo-0.5)*10k}
.param t1={td1/2} t2={td2/2} t3={PI*(1/2-fsw*(t1+t2))}
E1 11 22 VALUE={tanh((V(FB)-V(D))*10k)}
```

Conditional Evaluation of Expressions

Parameters or source values can be evaluated in expressions containing conditional statements.

Syntax

```
VALIF(CONDITION,expression1,expression2)
EVAL(CONDITION,expression1,expression2)
```

If CONDITION is TRUE, then VALIF (or EVAL) returns expression1 else it returns expression2. You can use the keyword VALIF (or EVAL) in any expression.

Conditional Evaluation Example:

```
.param v1=1.0
.param v2=2.0
.param v3=valif(v1>v2,v1+0.5,v2+0.5)
```

Here, v3 will be assigned the value 2.5. The EVAL syntax is closer to C language, which can be more convenient for some users.

Temperature Handling

ROHM Solution Simulator enables temperature handling using the commands .TEMP:, TNOM, TMOD and T and enables formulation of temperature dependent functions using the variable TEMPER.

The TNOM function from the .OPTION command is used to set the nominal simulation temperature, the temperature at which parameter calculations are made. The default temperature is 27 °C.

TNOM can appear in expressions.

TNOM is a reserved keyword, however it can be specified as a parameter in a .PARAM command when .OPTION DEFPTNOM is set. The temperature value that ROHM Solution Simulator uses is always that which is set with .OPTION TNOM=<*value*>.

Example 1: Setting TNOM with the .OPTION command

```
.option tnom = 100
```

Example 2: Setting TNOM with the .PARAM command

```
.option defptnom
.param tnom=100
v1 11 22 tnom
```

In this example, the voltage source uses the value specified in the .PARAM command tnom. If the DEFPTNOM option is not used, the default TNOM value is used.

The TMOD parameter (in certain models) sets the model temperature. The T parameter (in certain devices) sets the temperature of an individual instance of a device or model.

The order of priority of the above temperature related commands and parameters is T, then TMOD and then .TEMP, with decreasing priority. T has the highest priority.

TEMPER Variable

TEMPER is a variable returned by the simulator that gives the value of the current simulation temperature for use in subsequent calculations. This variable will be the present simulation temperature resulting from either a .TEMP command, a .DC TEMP sweep or, if neither are specified, the value of TNOM given in the .OPTION command. You can use the TEMPER variable in the formulation of temperature-dependent expressions. Any expressions containing the TEMPER variable will be automatically re-evaluated in the case of a change in this temperature.

Temperature Handling Example

You can use the TEMPER variable in conjunction with VALUE={Expression} in SPICE resistors, capacitors, and inductors to specify devices whose values vary with temperature.

The following example specifies a capacitor instance (C1) connected between nodes 11 and 22. Its value is calculated as the nominal capacitance C0 multiplied by (1+ 0.002 multiplied by the square of the current simulation temperature TEMPER). The TEMPER variable may also be used in expressions for model parameters.

```
C1 11 22 VALUE={C0*(1+0.002*(TEMPER^2))}
```

Physical constants

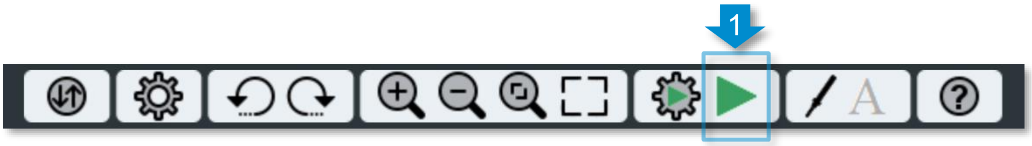
Since physical constants are not prepared in advance, it is convenient to write them as parameters of the .PARAM command when used.

Examples of major physical constants are shown in the table below (SI units).

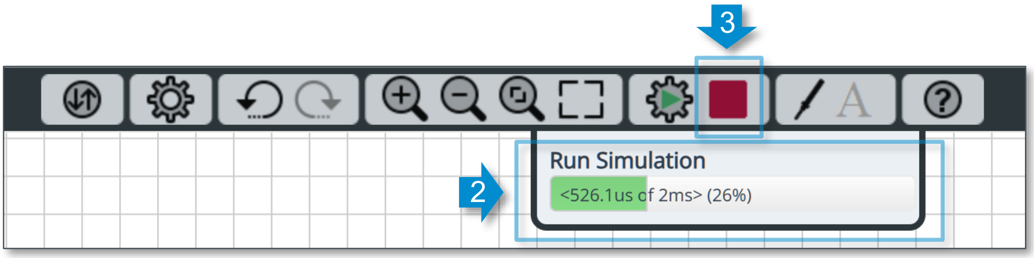
Major physical constants	Example description
Free space permittivity ε_0	`.param epso = 8.8541878188e-12`
Boltzmann constant k	`.param boltz = 1.380649e-23`
Electron charge e	`.param charge = 1.602176634e-19`
Absolute zero	`.param kelvin = -273.15`
Approximate value of pi	`.param pi = 3.14159265359`

10. Performing a Simulation

Click the right-pointing triangle icon for "Run Simulation" (1) to start a simulation.

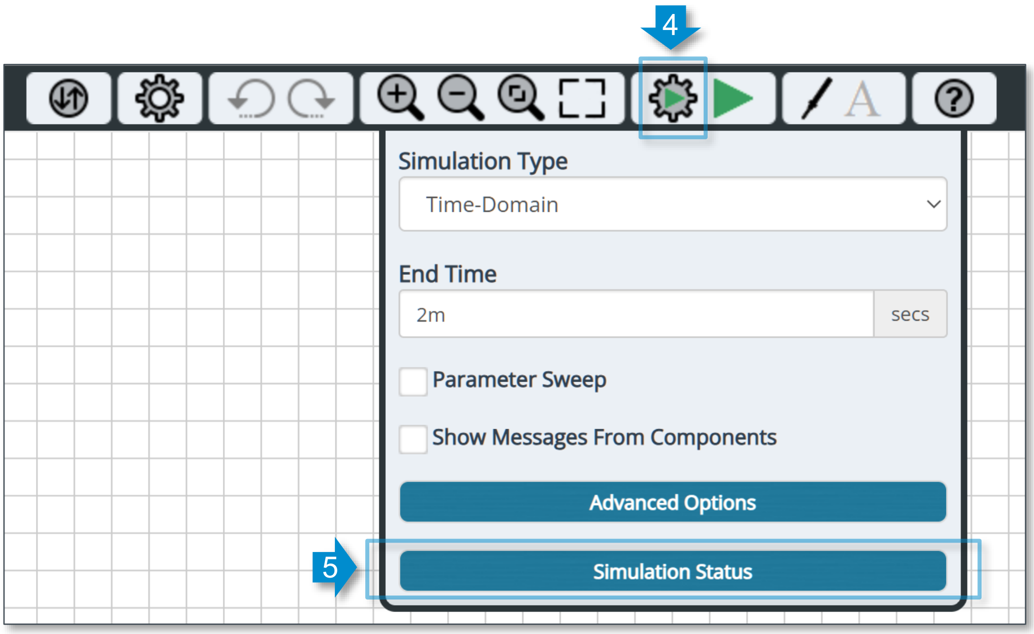


The progress is displayed under the tool bar while the simulation is running (2). The execution time of simulations is limited to 60 minutes for each run. The simulation is forced to terminate if 60 minutes elapses and the data up to the time point is displayed. To stop the simulation, click the red square icon for "Intercept Simulation" (3). Note that the simulation cannot be resumed once it is stopped.

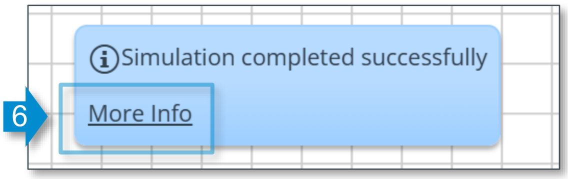


After the simulation is completed, the data is downloaded from the cloud server and the waveforms are displayed. However, it may take longer to download the data if the number of data points is larger.

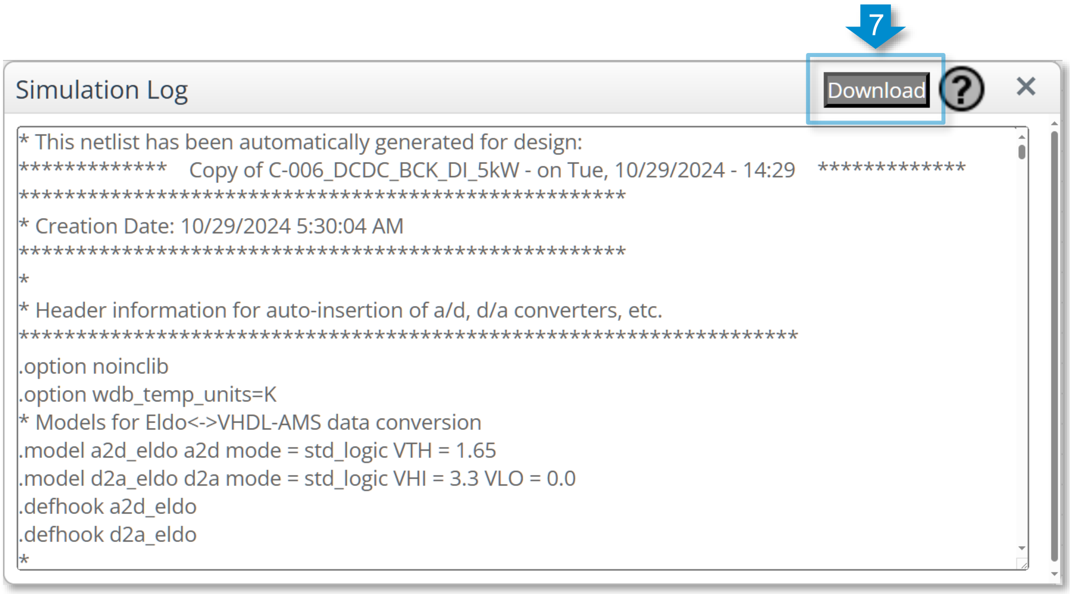
To view the simulation result, click the "Simulation Settings" icon (4) on the tool bar and then click "Simulation Status" (5) in the drop-down dialog.



The figure shown below is displayed if the simulation is completed normally. In addition, click “More Info” (6) to display “Simulation Log”.



The simulation log contains information such as the net list, simulation commands, simulation options, simulation log, DC voltage of the nodes, etc. Click the “Download” button (7) on the upper right to download the file to the local PC.



11. Waveform Operations

For the simulation result, the waveforms of the wiring and the inside of components can be displayed using a waveform viewer tool. Two waveform viewer tools are available as follows.

- Wavebox: → 11-1. Wavebox Tool

  Direct probing to the circuit diagram displays the waveforms inside a small window (box). Several waveforms can be checked easily. In addition, a minimum measurement function (cursor) is available.

- Waveform Viewer: → 11-2. Waveform Viewer Tool

  This is a sophisticated viewer with various measurement functions allowing you to display many waveforms in the same dialog and make comparisons with previous simulation results.

11-1. Wavebox Tool

Wavebox Display and Clear

Click the "Waveform Probe" icon (1) on the tool bar to display the Wavebox (2) and the probe (3). While controlling the probe with the mouse, move the probe over the wiring in the circuit diagram and then click the wiring to display the voltage waveform. In addition, you can also reposition the probe on another wiring location.

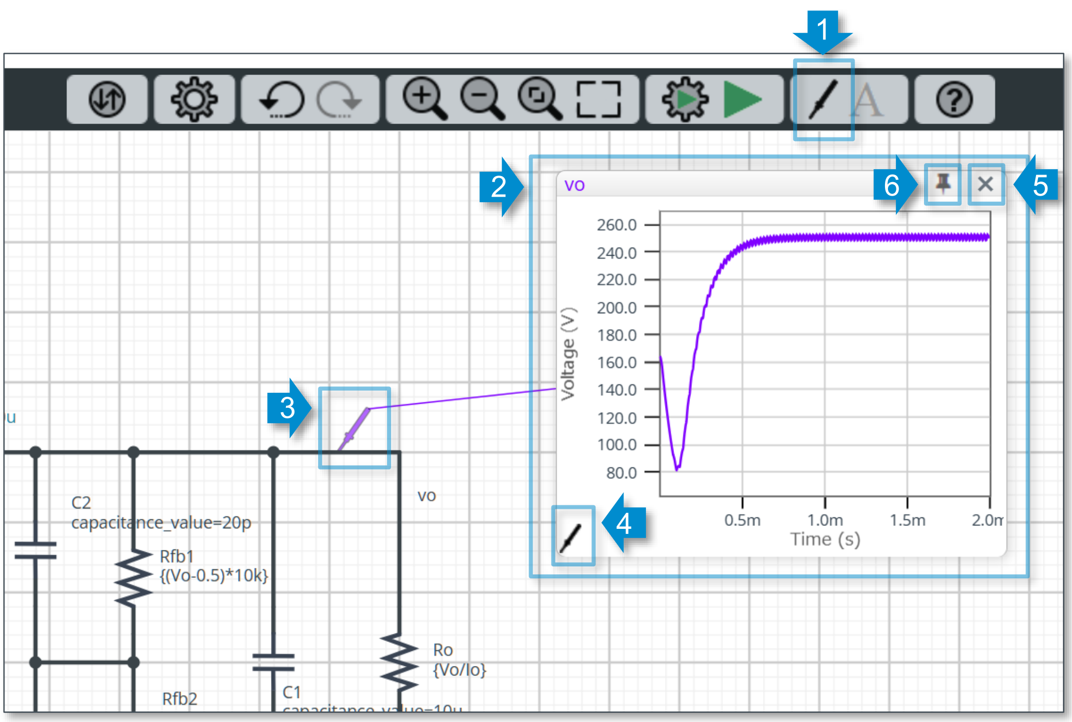
To add waveforms, drag and paste the probe icon (4) on the lower left of the Wavebox to display up to eight waveforms in the same Wavebox. To display waveforms in a new Wavebox, click the probe icon (1) on the tool bar.

Click the "Close" button (5) on the upper right if the Wavebox is unnecessary.

The Wavebox operation can be changed with the "Pin" icon (6) on the upper right as follows.

Vertical (default): The Wavebox size and position are adjusted according to the scale of the circuit diagram.

Horizontal: The Wavebox size and position are maintained even if the scale of the circuit diagram is changed.



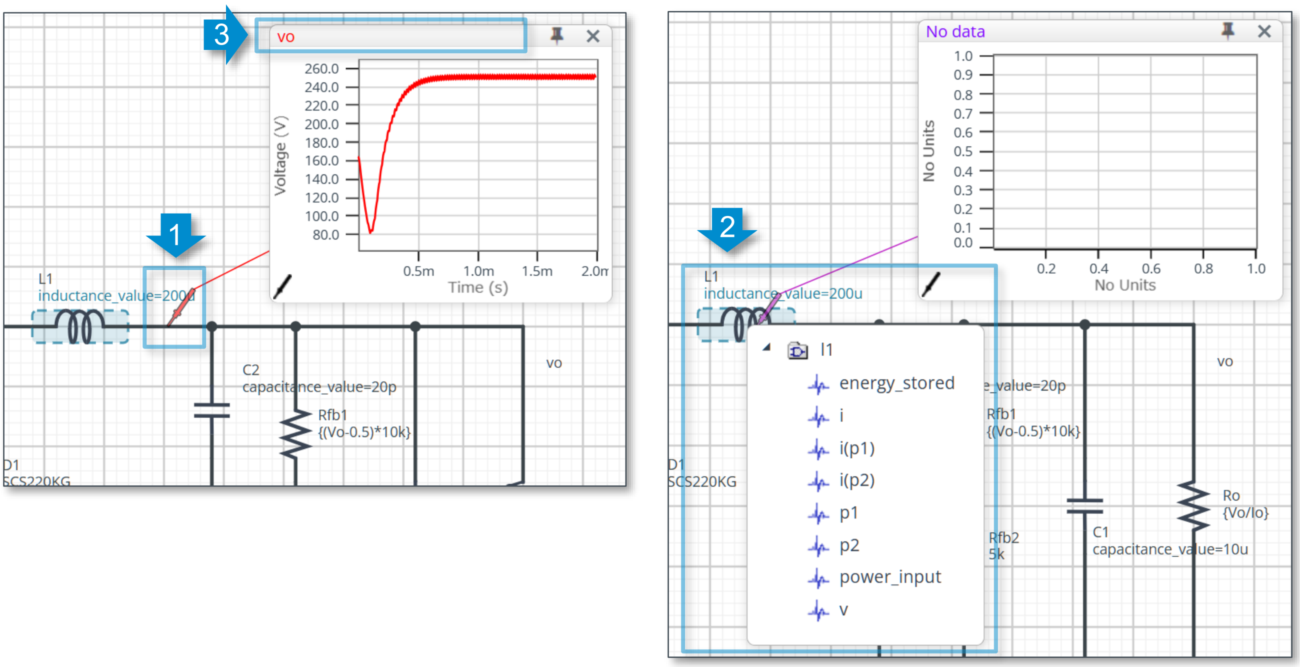
Plot a waveform

The target you touch with the probe determines the waveform that can be displayed as follows.

Wiring: A voltage waveform is displayed (1).

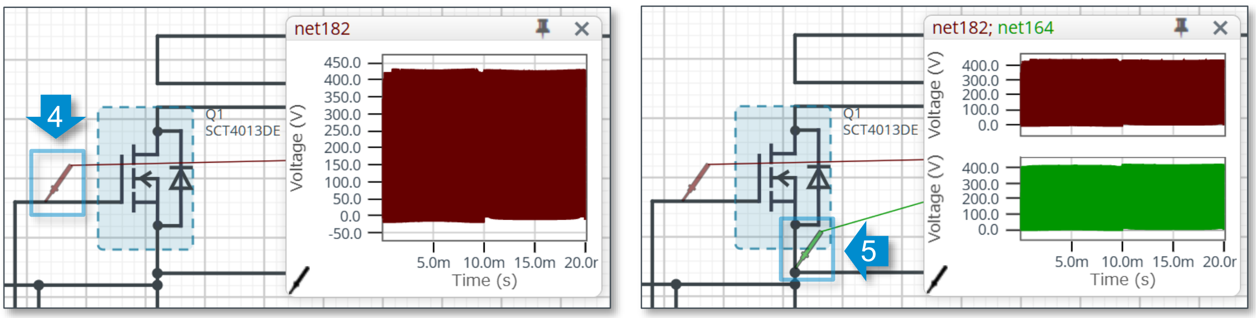
Component: Select a waveform in the waveform list displayed in the drop-down menu (2). Select i(xxxx) to measure the current that flows into the component.

The node name of the displayed waveform is shown at the top (3) in text in the same color as the waveform. The node names line up here if there are multiple waveforms.

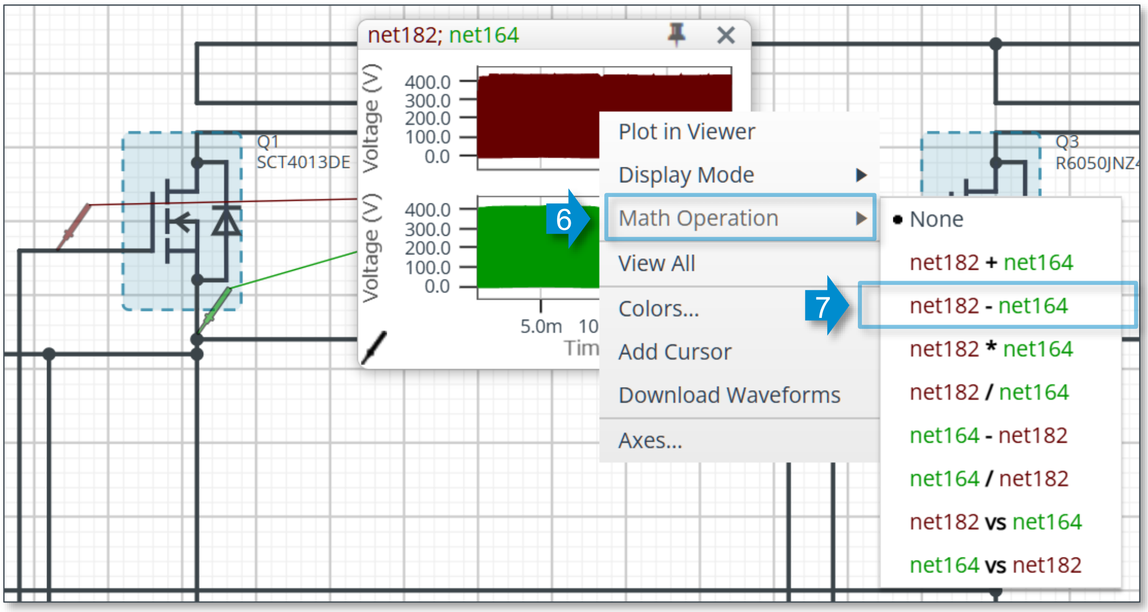


If the probe is placed normally as described above, the voltage waveform against the ground is displayed. To display the gate-source voltage waveform of a MOSFET, for example, use the following method.

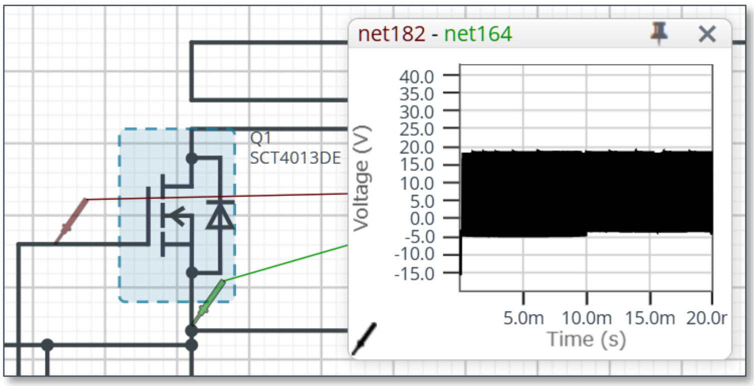
To display a voltage waveform between two points, use the Wavebox as a differential probe. First, place a probe on the gate (4). Next, place the second probe on the source (5).



Right-click on the Wavebox and then select "Math Operation" (6) in the context menu. When the list of calculation formulas is displayed, select a required formula (7).

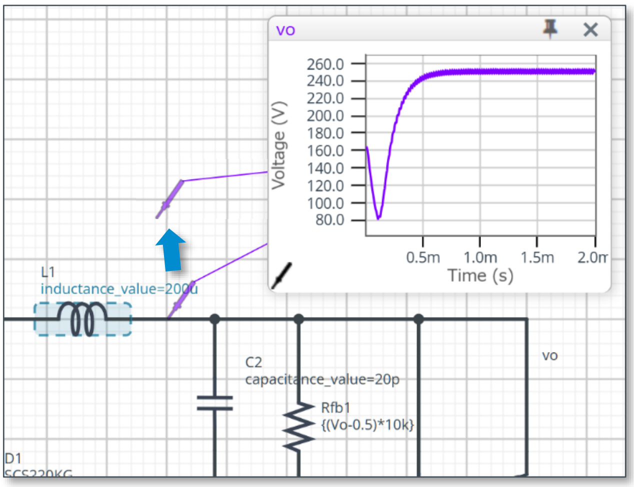


The voltage waveform between the two points is displayed (gate-source voltage in this example).



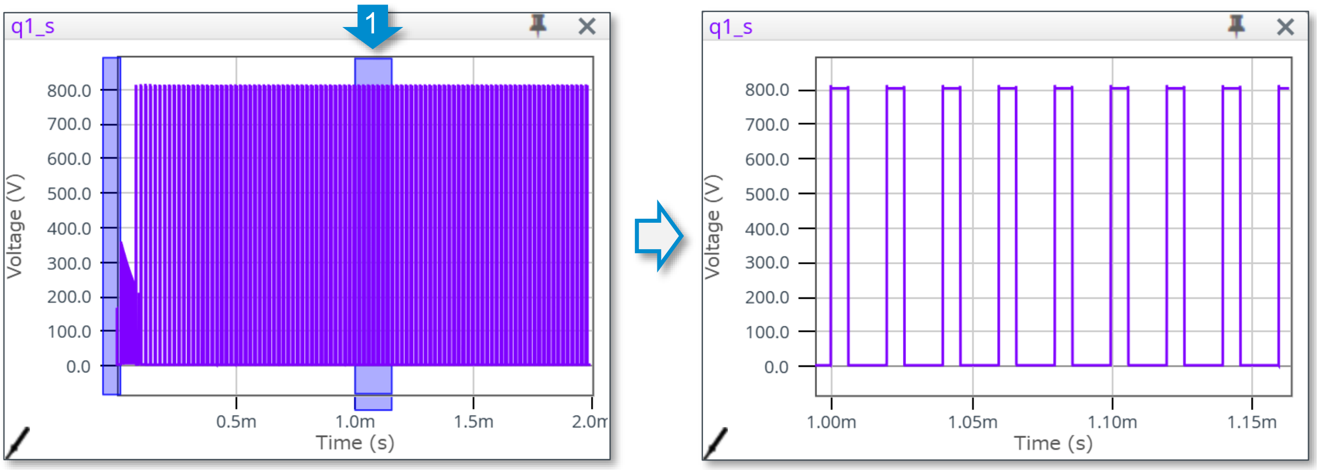
Delete Waveform

To delete the waveform, drag the probe and then drop it on an empty position.

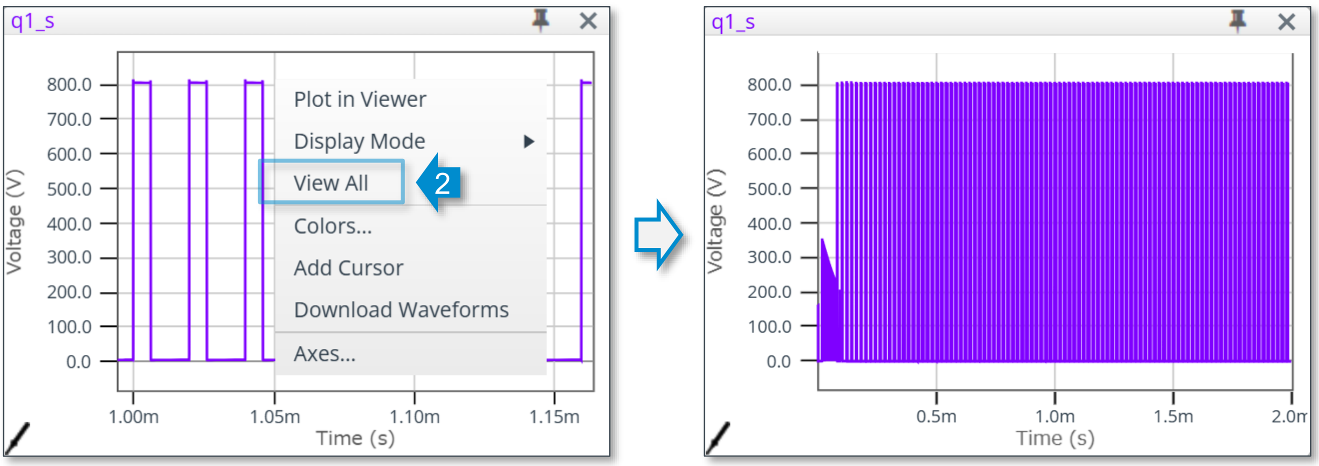


Zoom in/out on a Waveform Display

To zoom in on a waveform, press the left mouse button and drag the mouse to enclose an area to enlarge (1). The waveform is enlarged as shown in the right figure below.



To zoom out on a waveform, right-click on the Wavebox and then select "View All" (2) in the context menu. The entire waveform is displayed as shown in the left figure below.



Display Mode

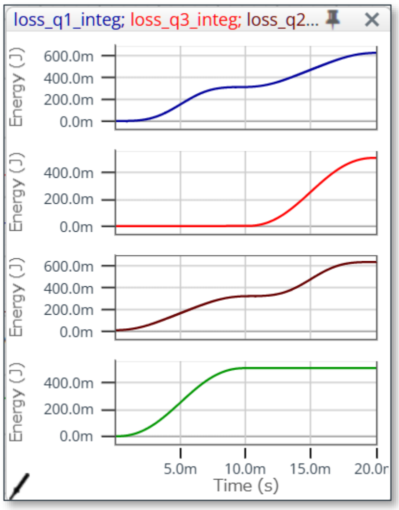
Two modes are available for displaying multiple waveforms as follows.

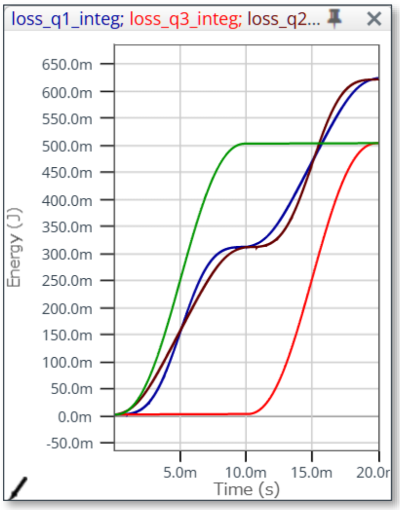
- Multi Trace (default)

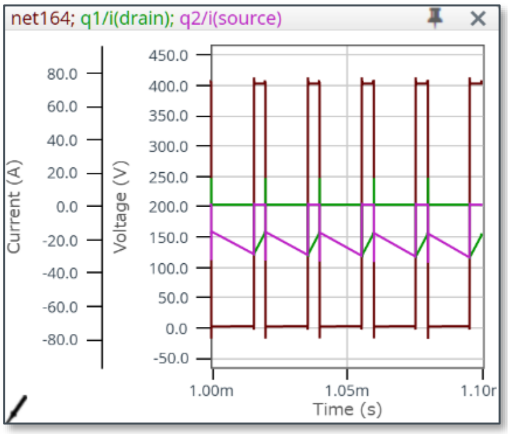
    Multiple graphs with one Y-axis for each waveform are vertically aligned (the left figure below).

- Overlaid

    Multiple waveforms are overlaid on one graph (the middle figure below). Multiple Y-axes are displayed if the units of the Y-axes are different (the right figure below).



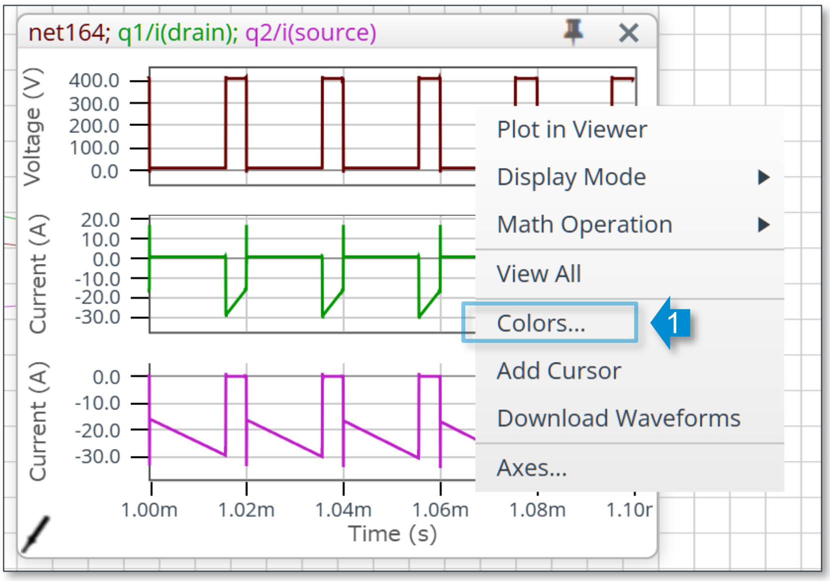




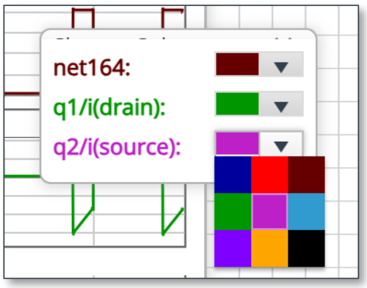
Waveforms Color

You can change the waveform color.

Right-click on the Wavebox and then select "Colors..." (1) in the context menu.



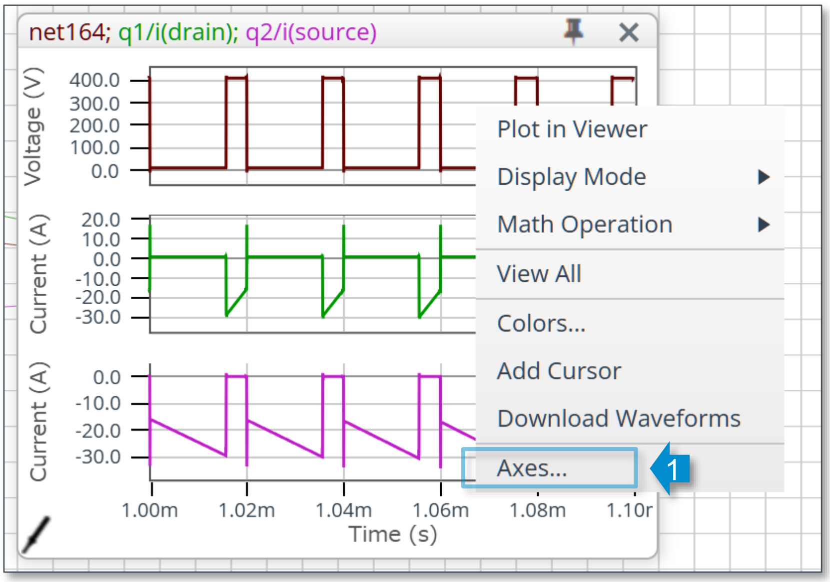
When the list of waveform names is displayed, select a color with the color palette.



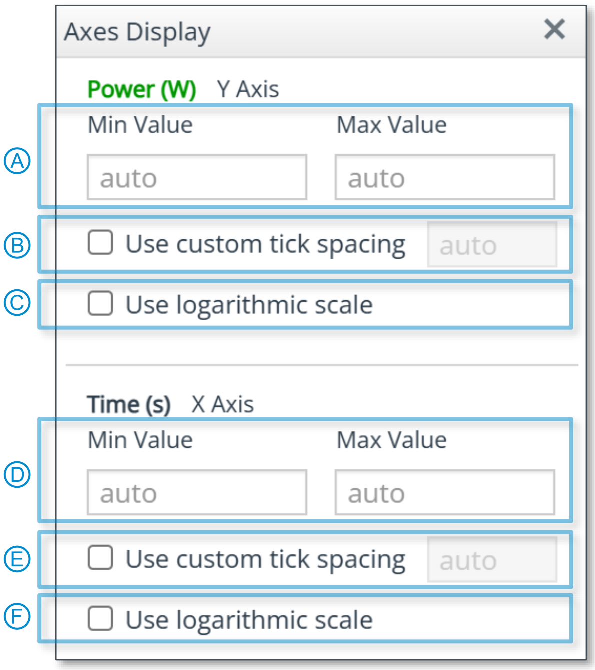
Waveform Axis

The X- and Y-axes of a waveform graph are usually scaled automatically. However, users can display the graph with a desired scale.

Right-click on the Wavebox and then select "Axes..." (1) in the context menu.



When the dialog is displayed, set the parameters you wish to change.



Y-axis

Ⓐ Enter numerical values to specify the minimum and maximum values of the Y-axis.Delete the input values to restore the automatic scaling.

Ⓑ To specify the tick spacing of the Y-axis, check the check box and enter a numerical value for the tick spacing. Uncheck the check box to restore the automatic scaling.

Ⓒ Check the check box to display the Y-axis on a logarithmic scale.

X-axis

Ⓓ Enter numerical values to specify the minimum and maximum values of the X-axis.Delete the input values to restore the automatic scaling.

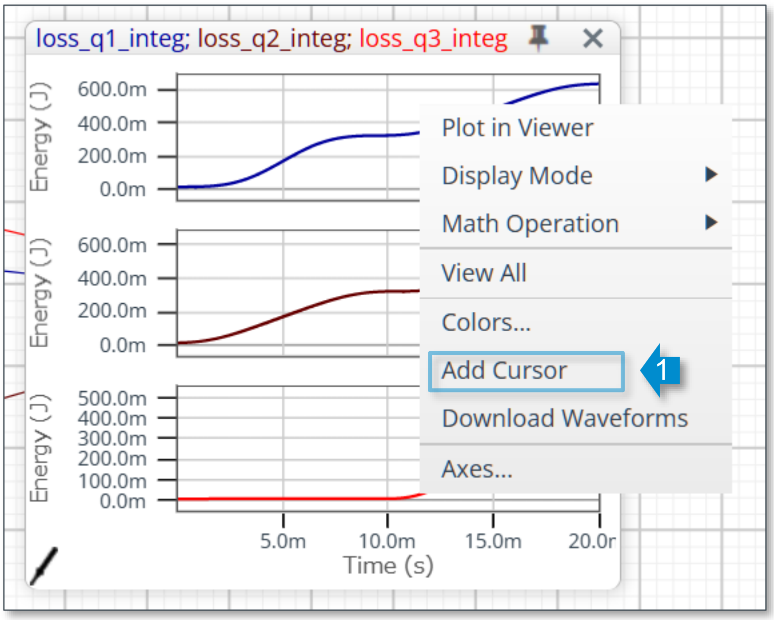
Ⓔ To specify the tick spacing of the X-axis, check the check box and enter a numerical value for the tick spacing. Uncheck the check box to restore the automatic scaling.

Ⓕ Check the check box to display the X-axis on a logarithmic scale.

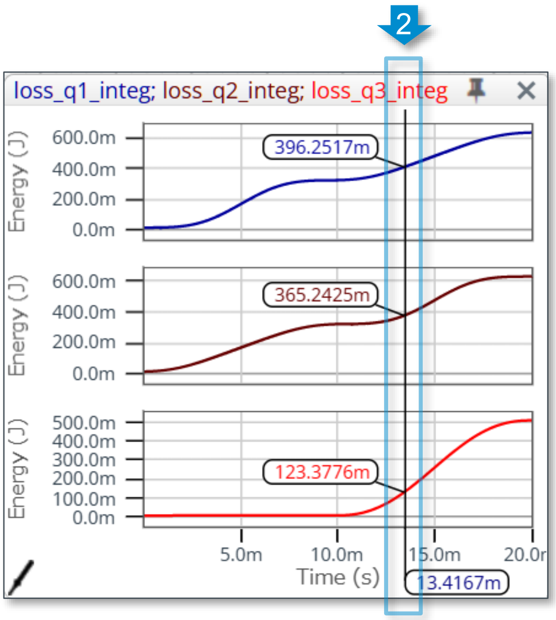
Waveform Cursors

You can display a cursor to read numerical values from waveforms.

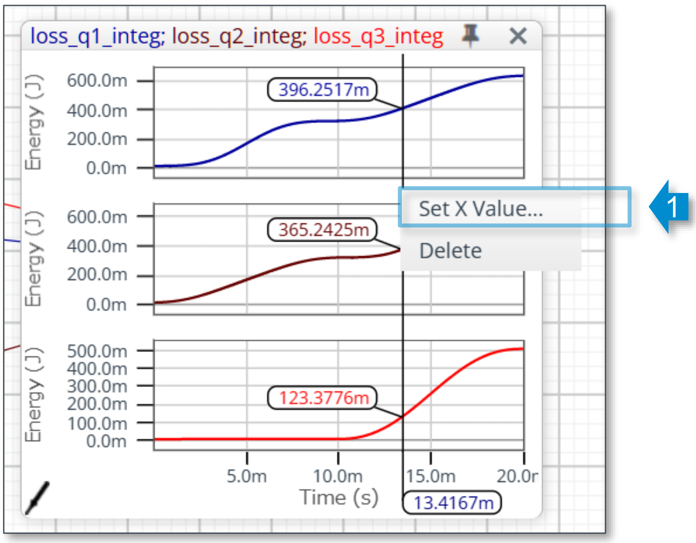
Right-click on the Wavebox and then select "Add Cursor" (1) in the context menu.

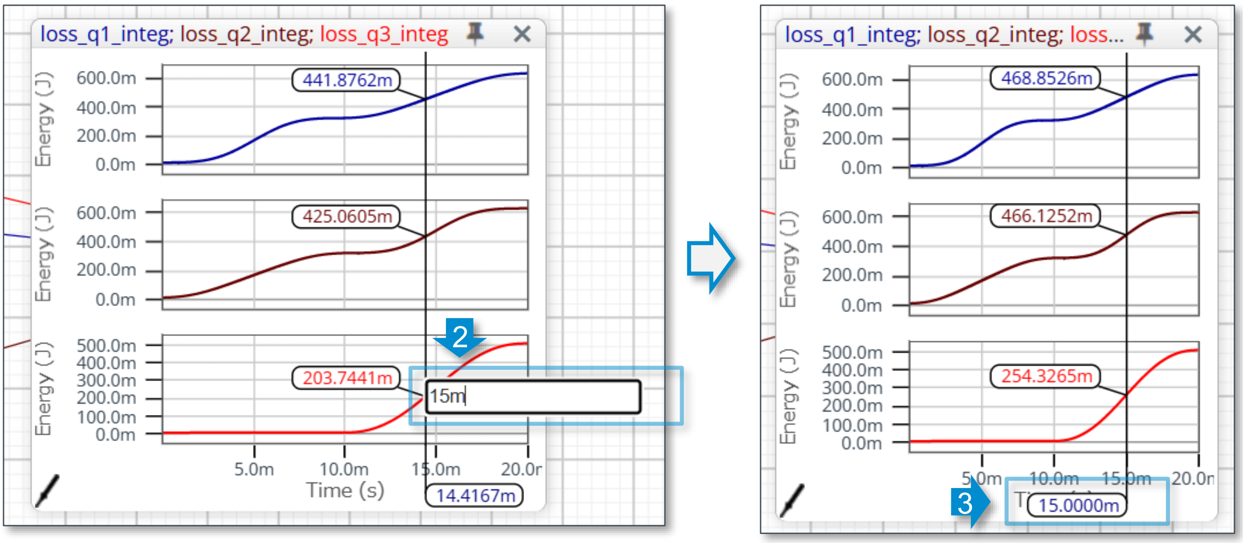


A cursor (2) is added, displaying the numerical values on the X- and Y-axes. You can drag and move the cursor to the left and right.

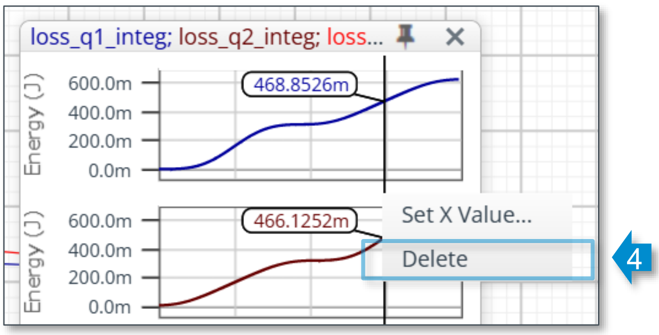


To move the cursor to a specified X-axis value, right-click on the cursor and then select "Set X Value..." (1) in the context menu. When the input field appears, enter a numerical value on the X-axis (2). The cursor is moved to the desired position (3).



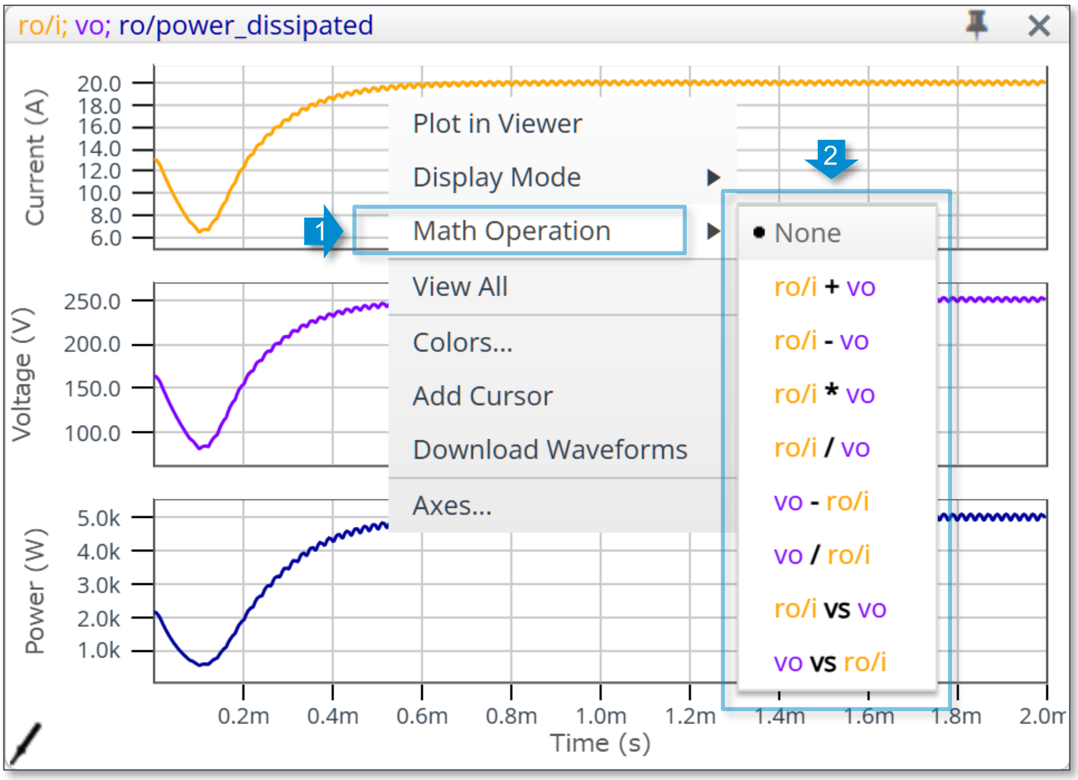


To delete an unnecessary cursor, right-click on the cursor and then select "Delete" (4) in the context menu. Note that you may not be able to delete the cursor if it is displayed initially for a circuit diagram provided by ROHM.

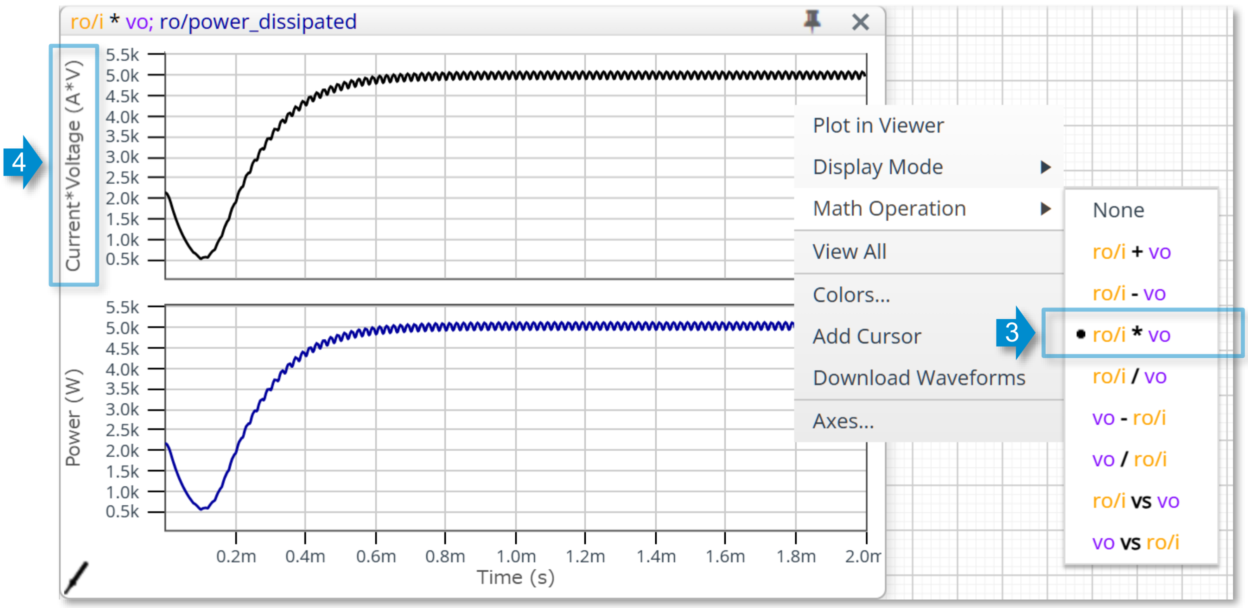


Mathematical Operation

You can perform simple calculations of waveforms. Right-click on the Wavebox and then select "Math Operation" (1) in the context menu to display the drop-down list (2) further on the right. Simple calculation formulas are automatically displayed here (cannot be changed). If multiple waveforms are displayed, the top two signal names are selected. Since the freedom of use is limited, basically use the Waveform Viewer and the Waveform Analyzer described below.



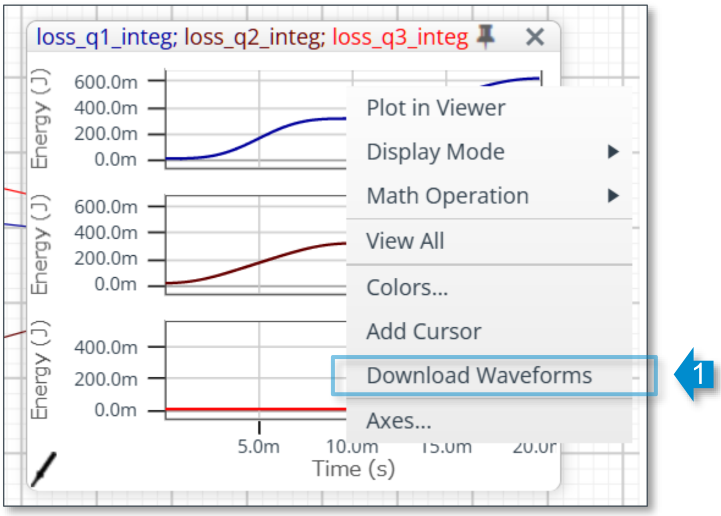
For example, calculating a product of a current and voltage (3) displays a power waveform (4).



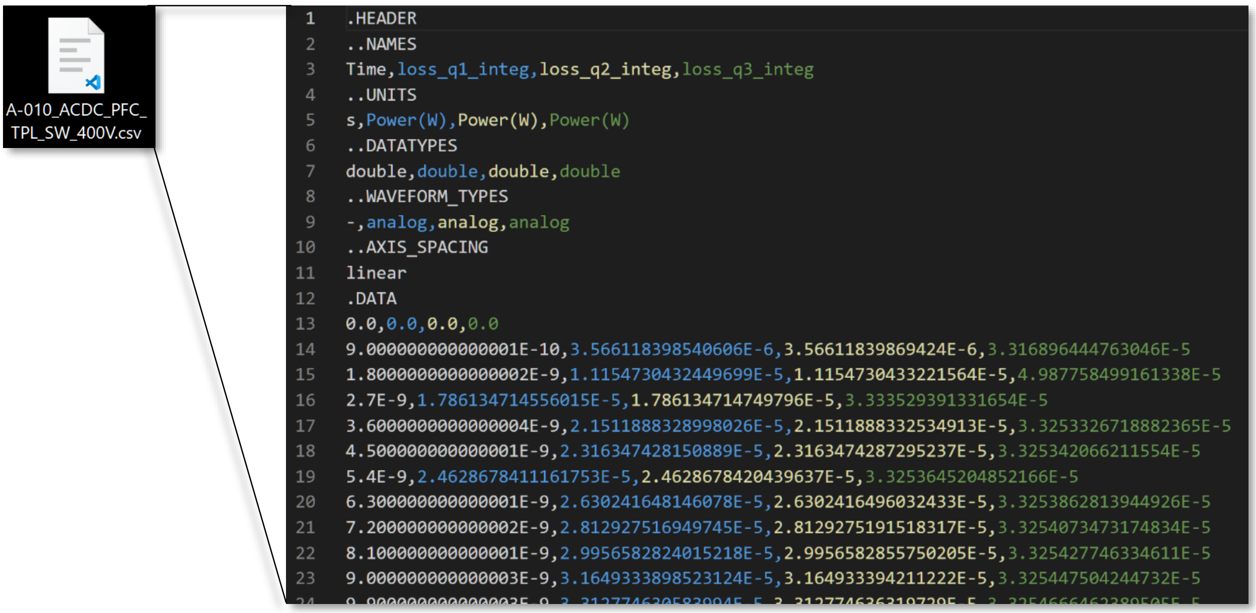
Download Waveforms

You can download the waveform data as a CSV file and utilize it with other software.

Right-click on the Wavebox and then select "Download Waveforms" (1) in the context menu.



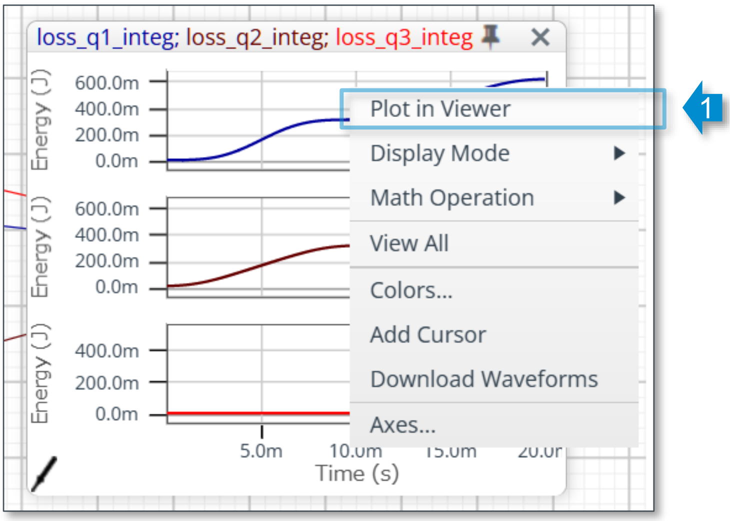
Wait for the data to be processed on the server and the download to complete. Note that it may take several tens of seconds to several minutes if there are many data points.

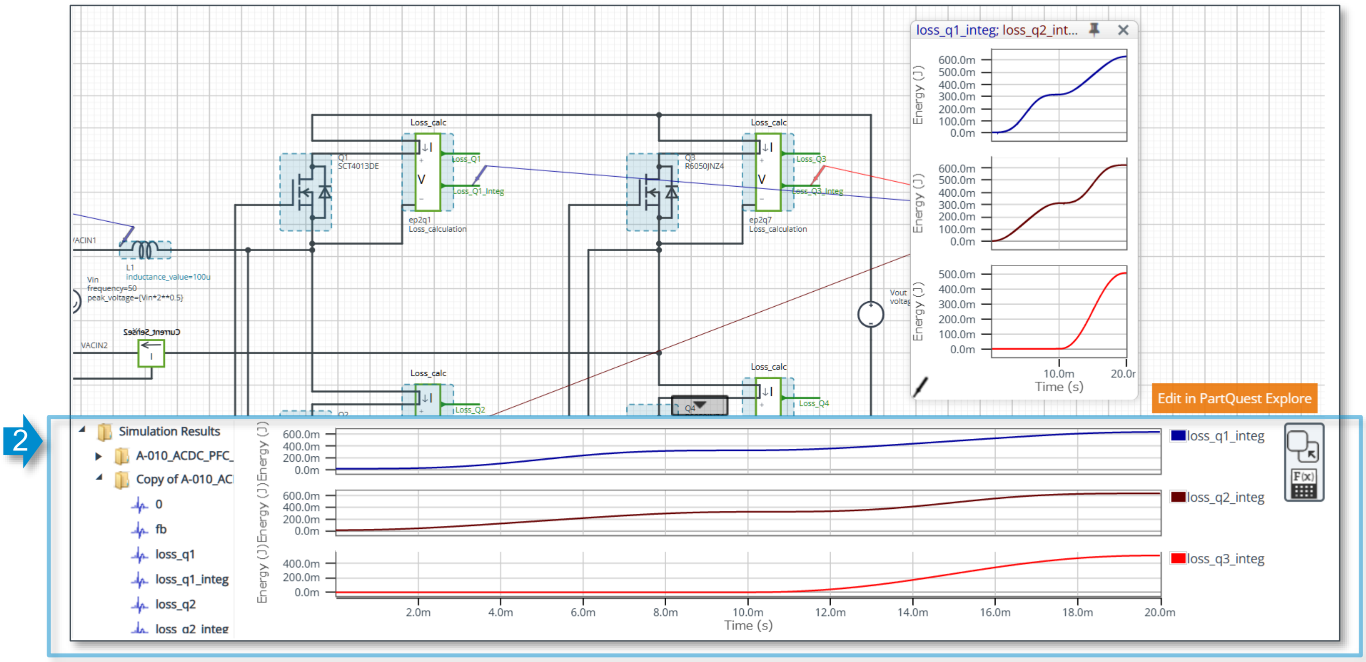


Plot in Waveform Viewer

You can copy the waveform data from the Wavebox to the Waveform Viewer for advanced waveform display and measurement.

Right-click on the Wavebox and then select "Plot in Viewer" (1) in the context menu. The Waveform Viewer is displayed in the bottom of the screen (2). See the section on the Waveform Viewer for how to use the viewer.

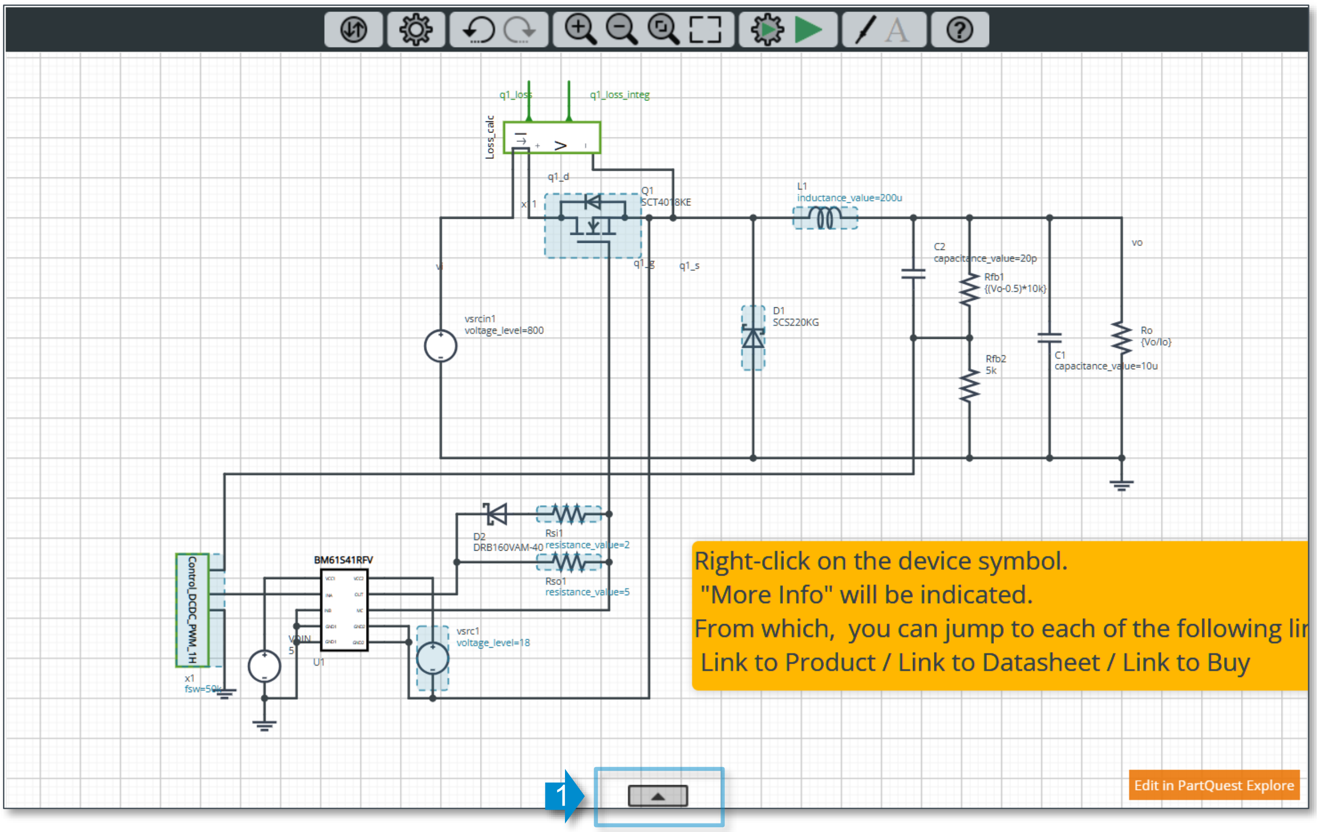


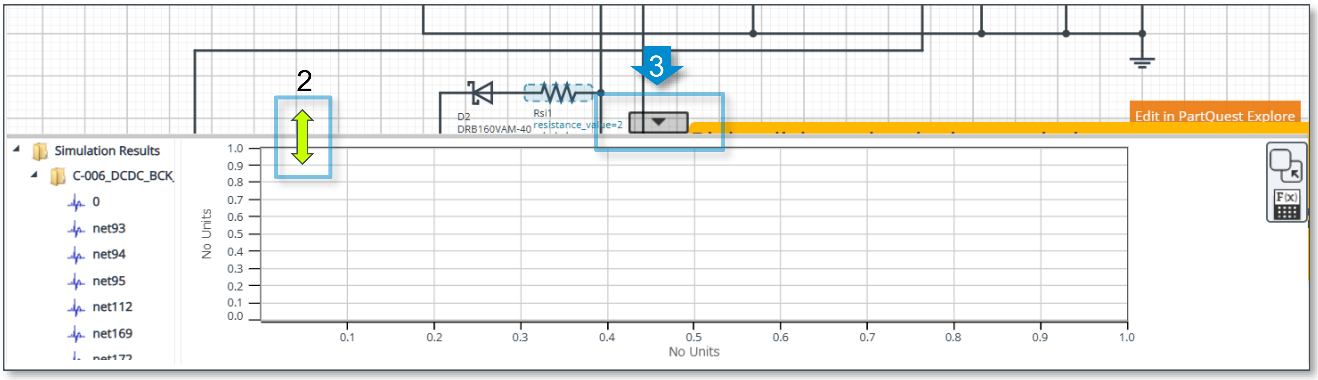


11-2. Waveform Viewer Tool

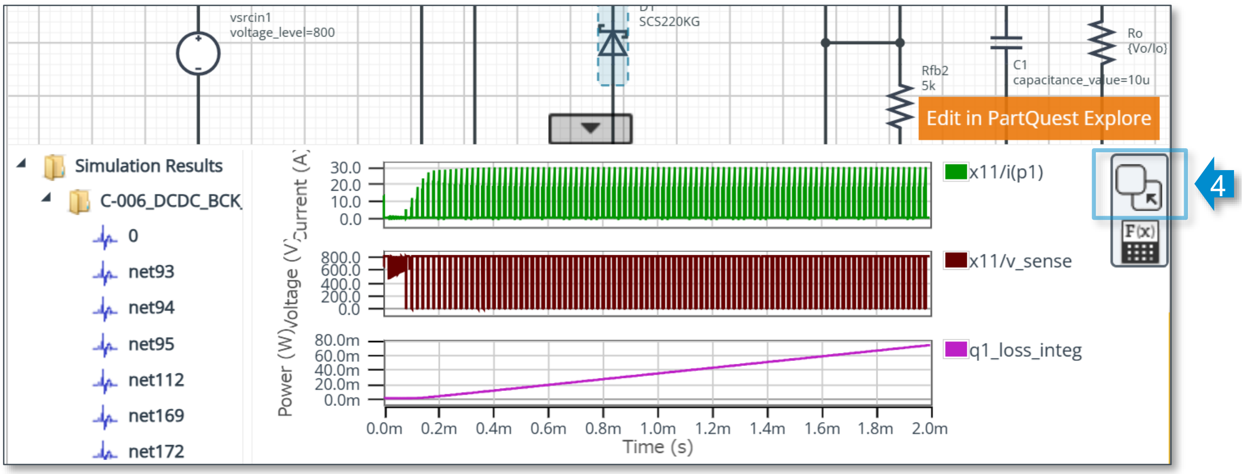
Displaying the Waveform Viewer

Click the triangle button (1) on the lower center of the screen to display the Waveform Viewer. To increase the display area, hover the mouse cursor over the border of the Waveform Viewer and drag it upward (2). To hide the Waveform Viewer, click the downward-pointing triangle button (3).

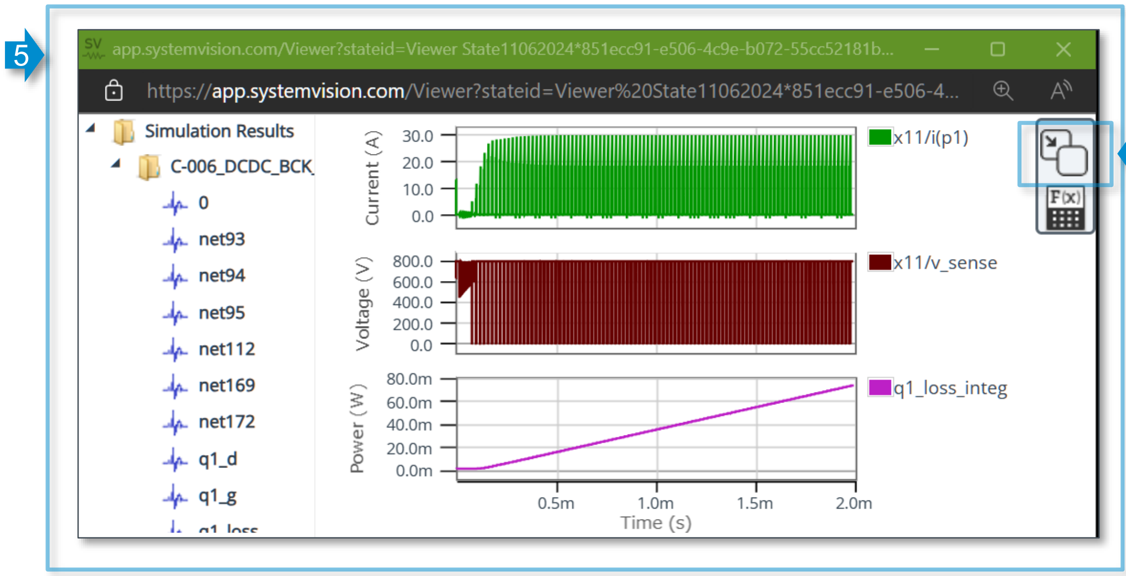




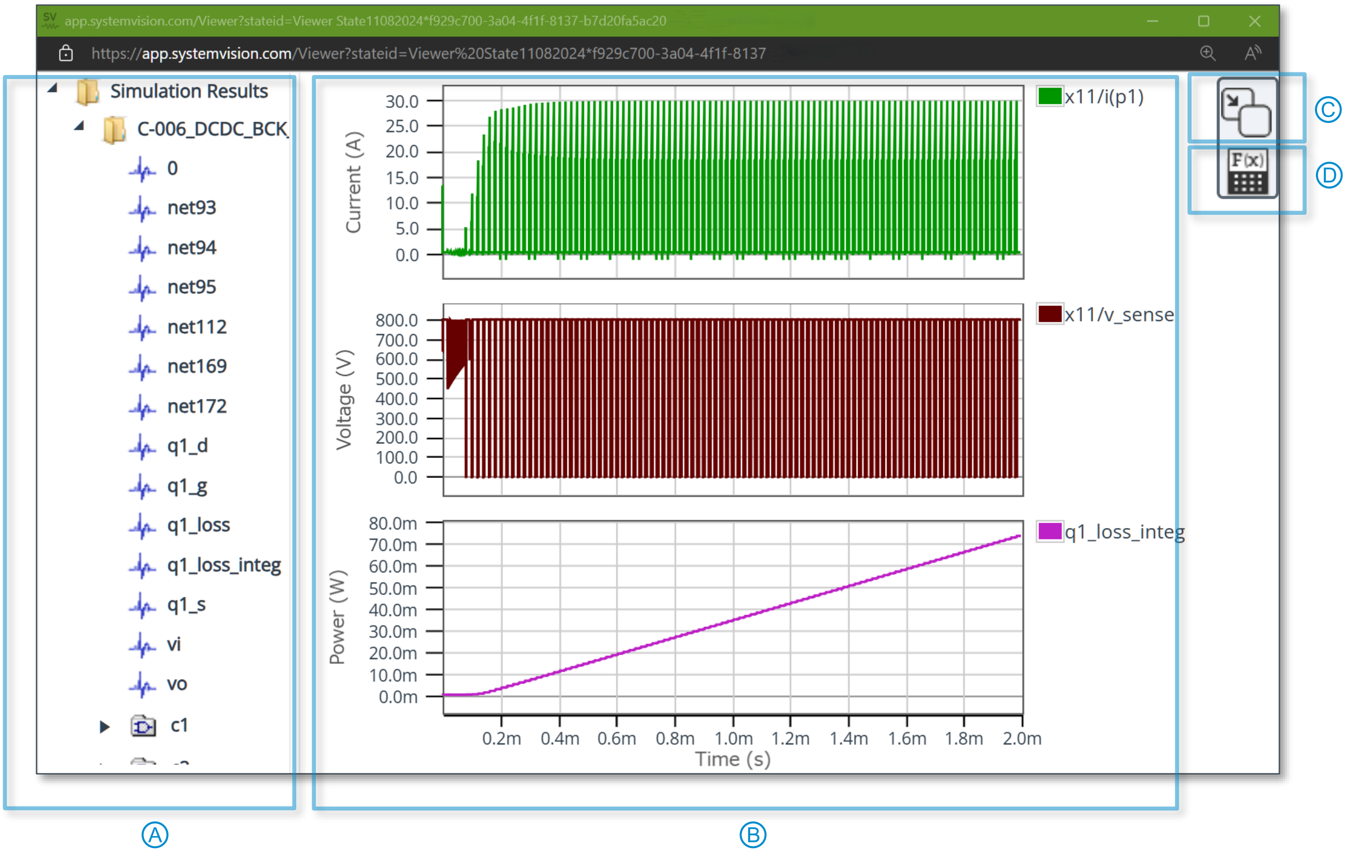
Clicking the icon (4) on the upper right of the Waveform Viewer displays the viewer in a new window for improved operability (5).



To pin the Viewer to the circuit diagram again, click the icon (6) on the upper right. Note that, if you click the close button (× mark) of the window inadvertently, the waveform display disappears and the Waveform Viewer cannot be started the next time (this specification will be improved).



Elements of the Waveform Viewer



Ⓐ Simulation result display area

Displays the list of internal signals for the circuit wiring and components.

Ⓑ Waveform display area

Displays the waveforms of the simulation results and the waveform measurement results.

Ⓒ Pin button

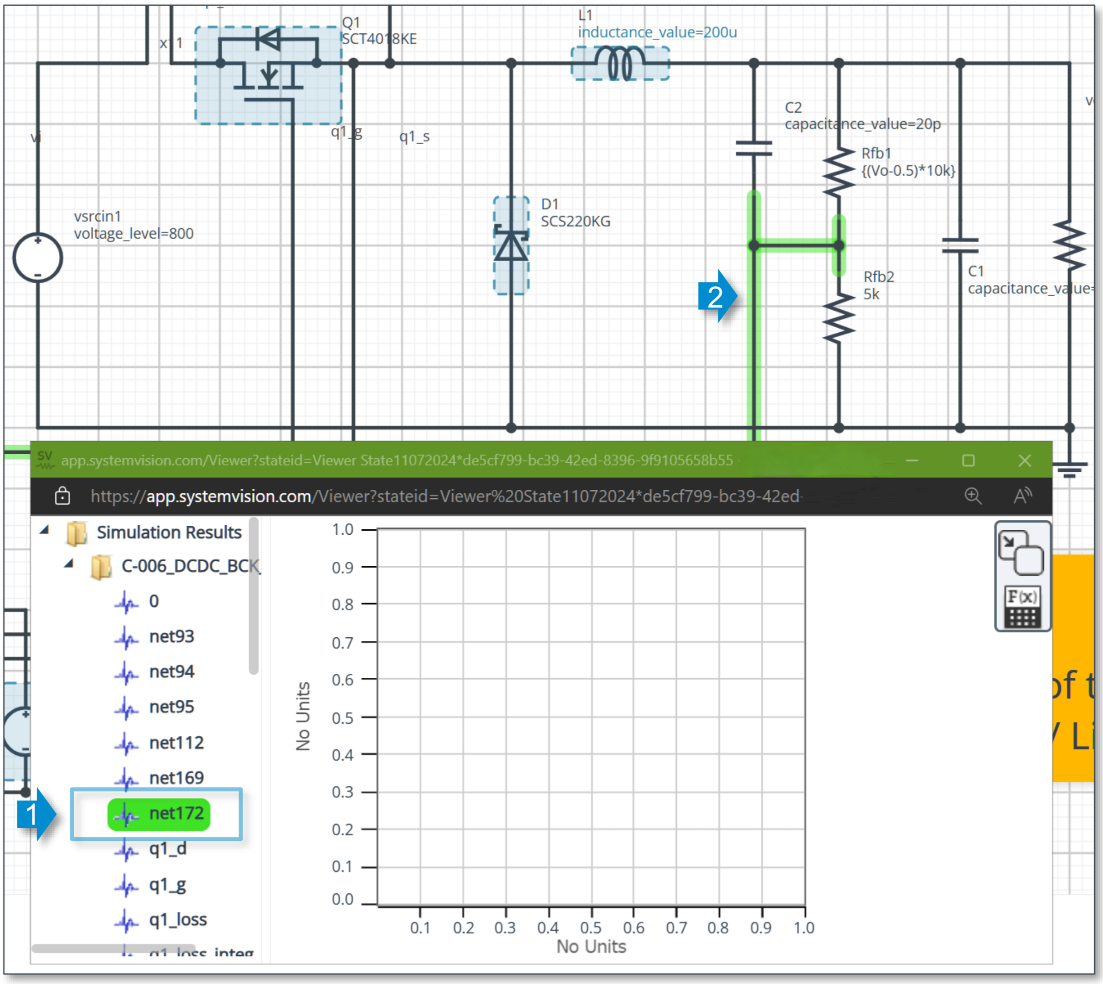
Pins this Waveform Viewer to the circuit diagram.

Ⓓ Waveform Analyzer

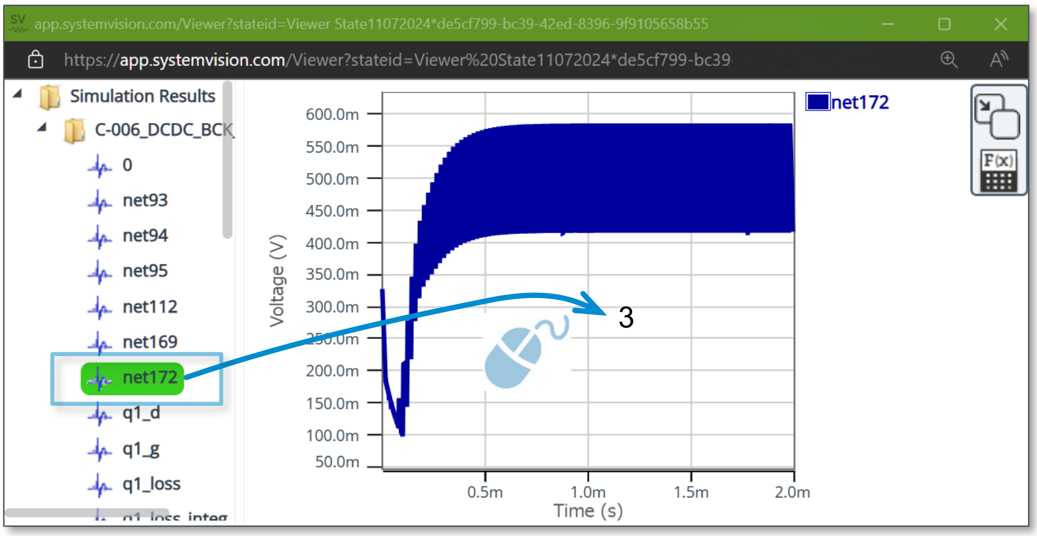
This tool is equipped with various measurement and calculation functions.

Plot a waveform

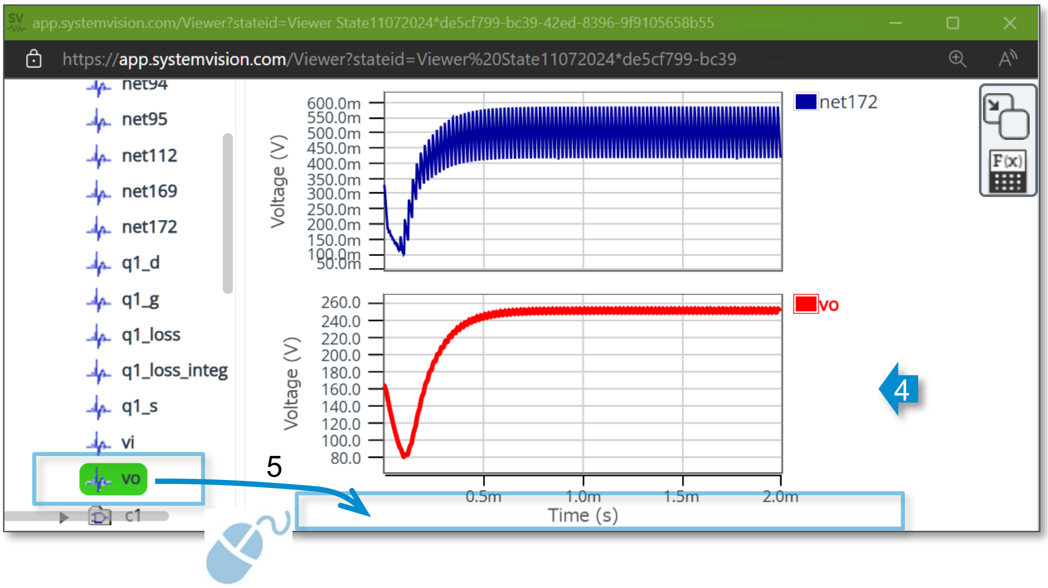
Hovering the mouse cursor over a net or component name in the simulation result display area (1) highlights the corresponding section (2) in the circuit diagram.



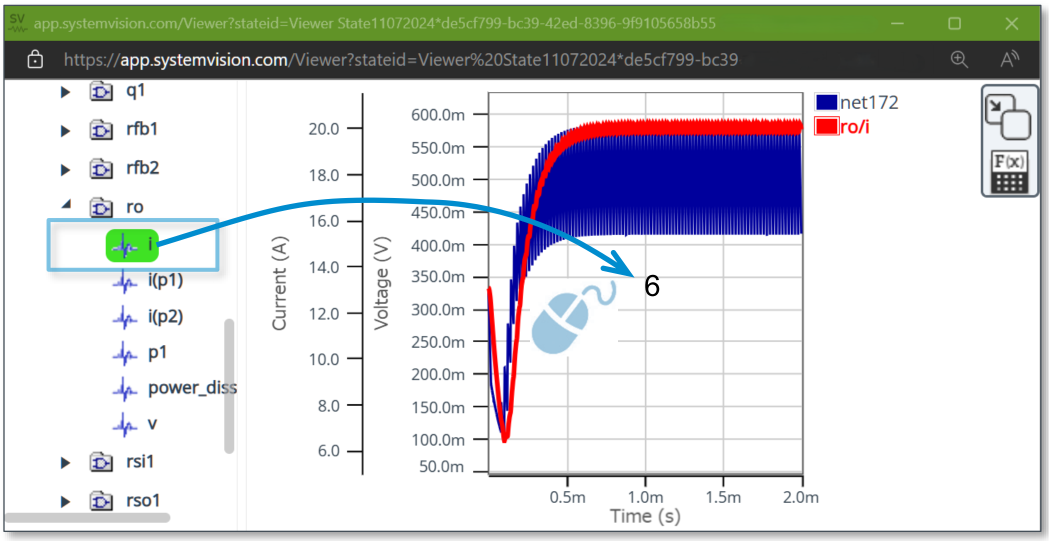
Double-click a net name or drag and paste it into the waveform display area (3) to display a waveform.



Double-clicking another net name adds a new graph at the bottom (4). To use the drag and paste method for adding a graph, drop a net name into an area at the bottom outside other graphs (5).

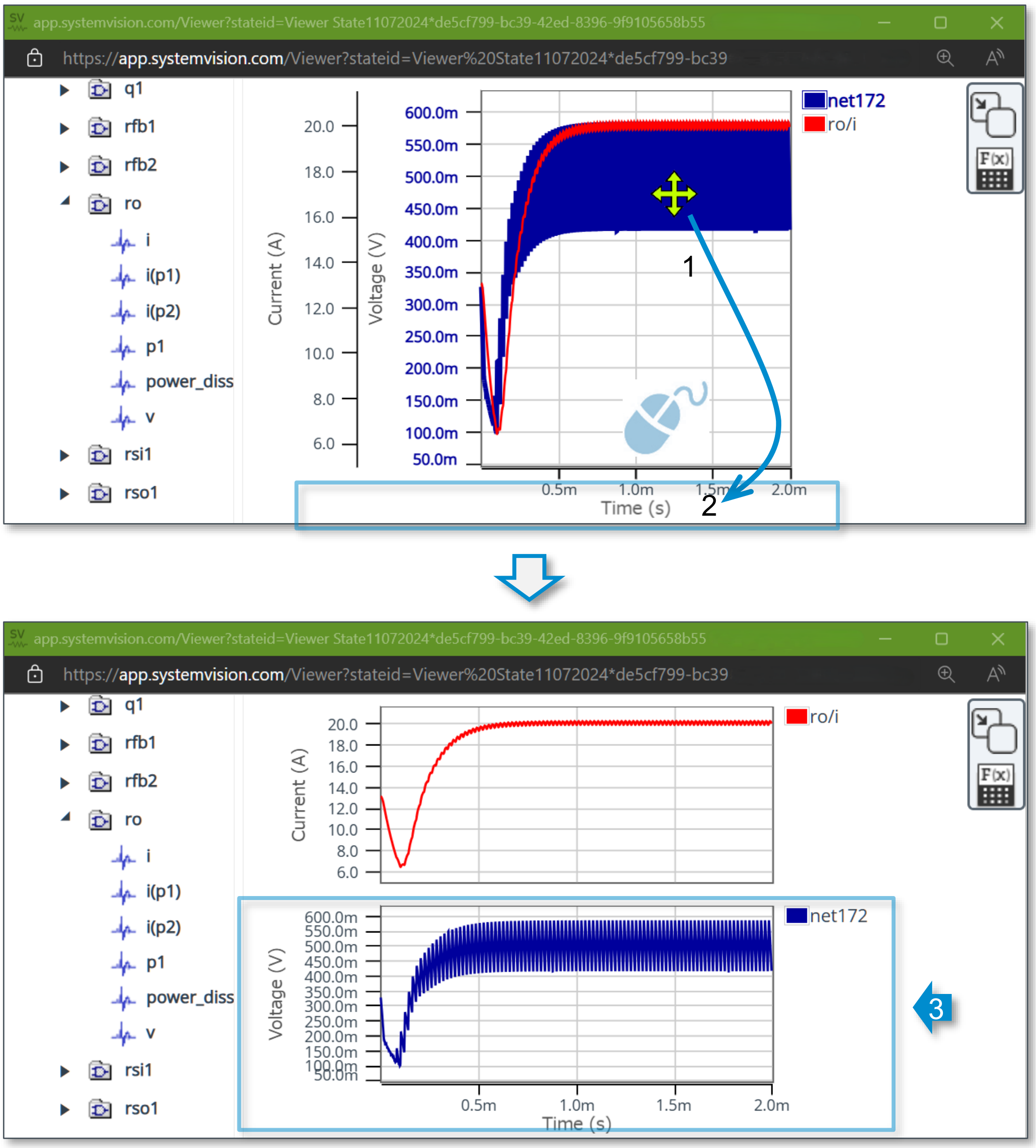


To overlay waveforms in the same graph, drag a net name into the graph (6).



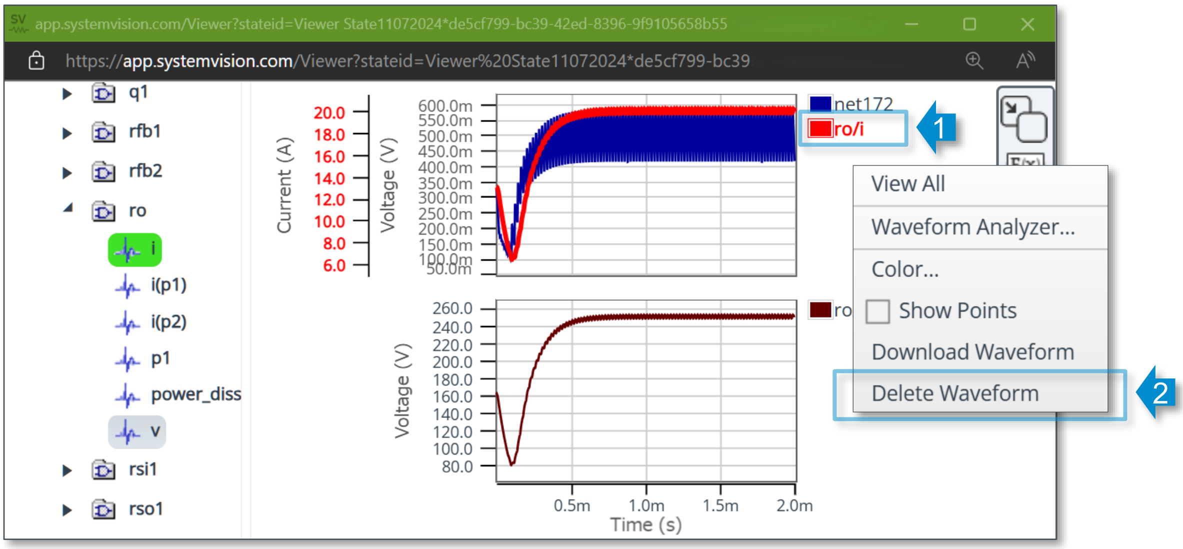
Waveform Window Operations

From a graph with overlaid waveforms, dragging a desired waveform (1) and dropping it outside the graph (2) can create another graph (3). Conversely, dragging a waveform from a graph and dropping it into a desired graph allows you to overlay waveforms in the graph.

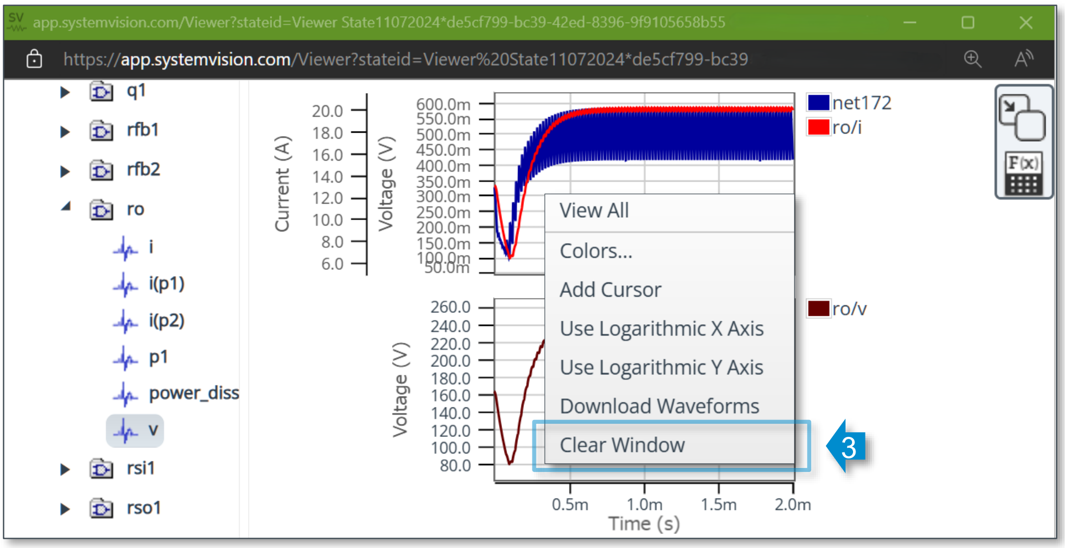


Delete Waveform

To delete an unnecessary waveform, right-click on the net name (1) displayed on the right of the graph and then select "Delete Waveform" (2) in the context menu. To delete a certain graph, deleting all waveforms displayed in the graph deletes the graph as well.

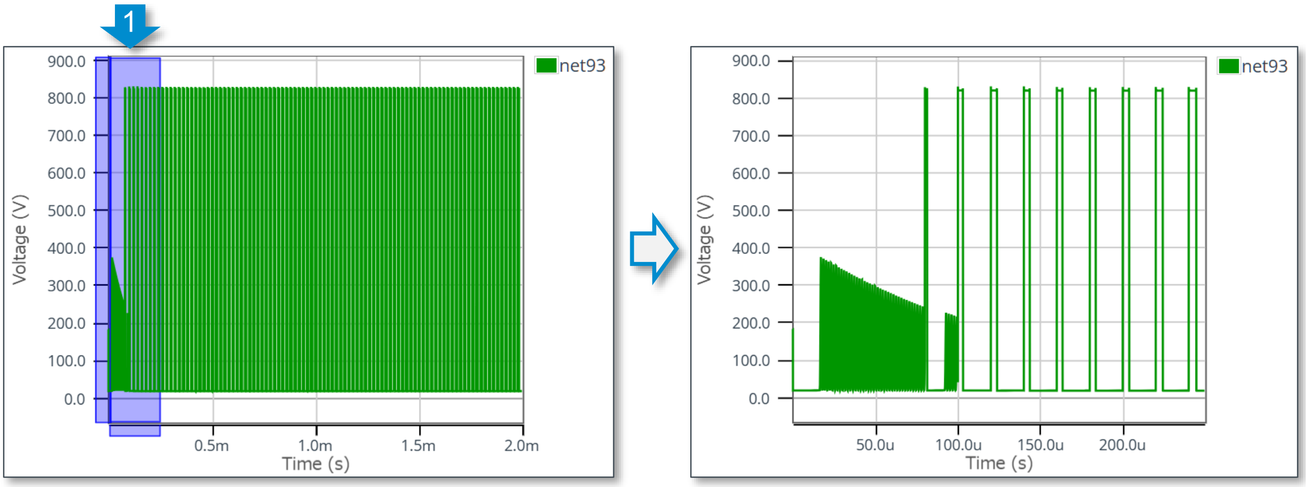


To delete all graphs and waveforms, right-click on any graph and then select "Clear Window" (3) in the context menu.

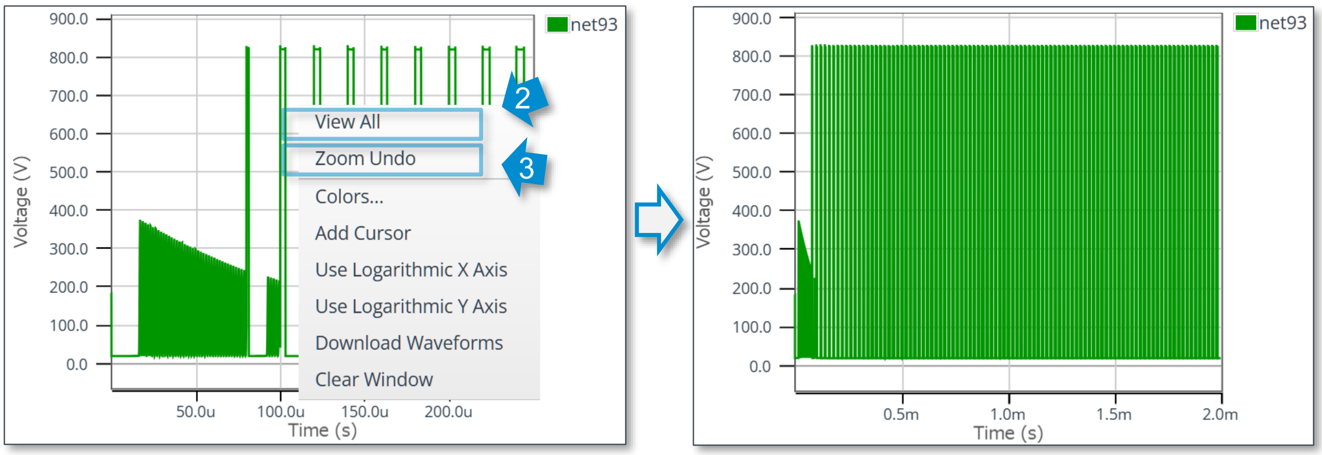


Zoom in/out on a Waveform Display

To zoom in on a waveform, press the left mouse button and drag the mouse to enclose an area to enlarge (1). The waveform is enlarged as shown in the right figure below.



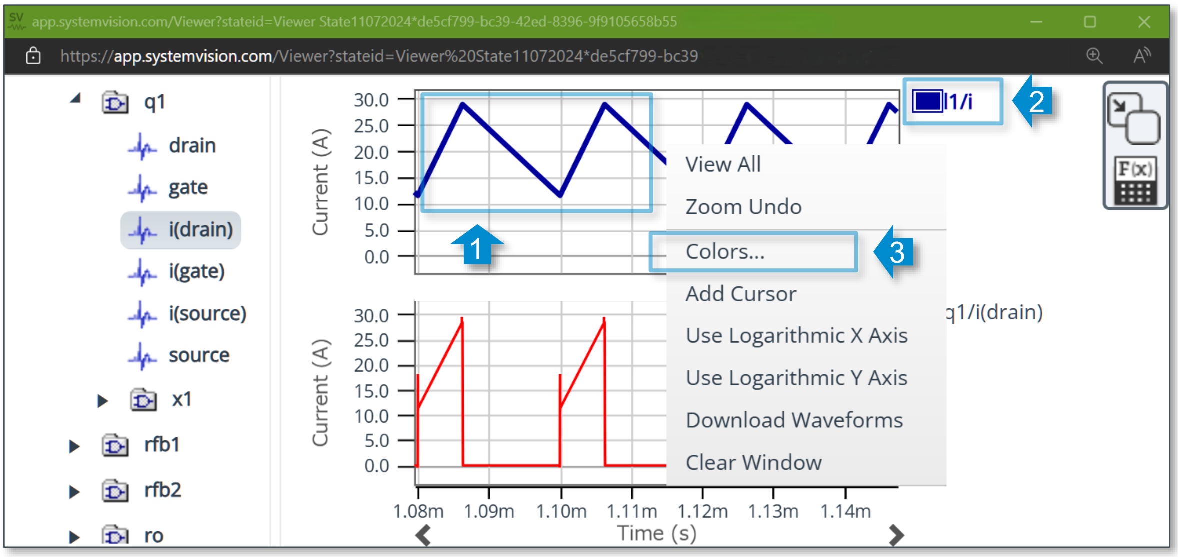
To zoom out on a waveform, right-click on the graph and then select "View All" (2) in the context menu. The entire waveform is displayed as shown in the left figure below. Selecting "Zoom Undo" (3) restores the previous display scale.



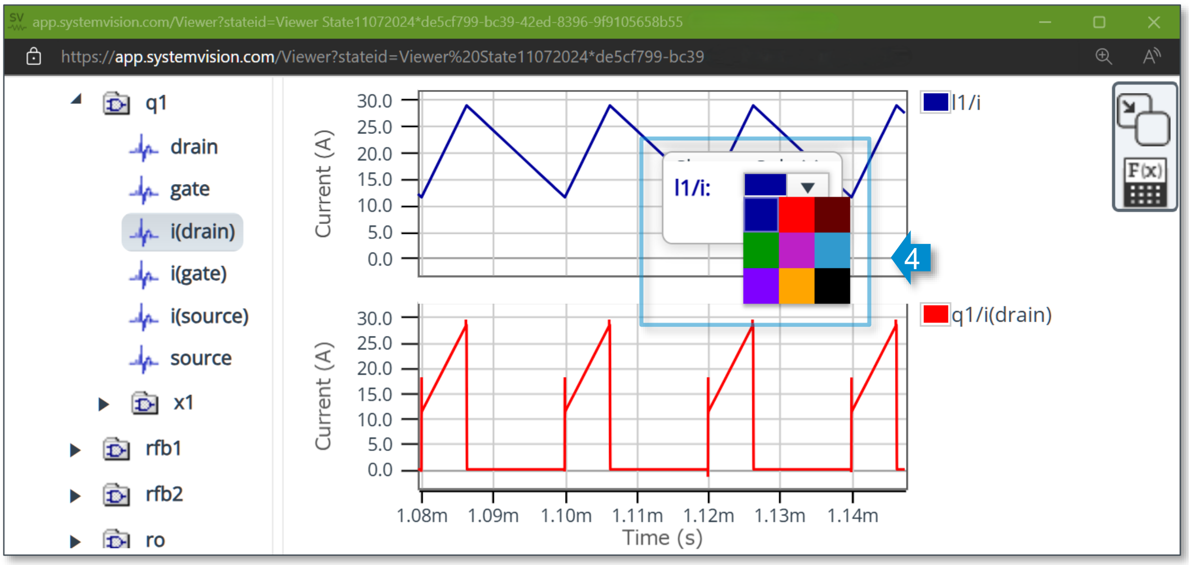
Waveforms Color

You can change the waveform color.

To change the color of a waveform, right-click on the waveform (1) or net name (2) displayed on the right of the graph and then select “Colors...” (3) in the context menu.



Select a color with the color palette displayed (4).

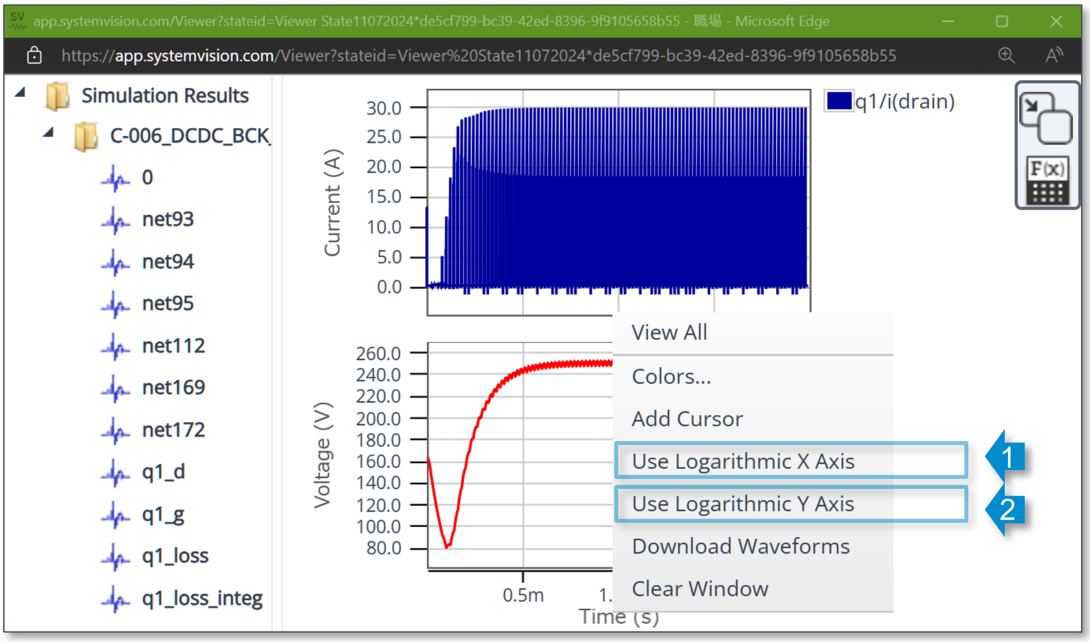


Waveform Axis

The X- and Y-axes of waveform graphs are automatically scaled and users cannot specify values as desired. However, the scale type can be switched to logarithmic.

To switch the X-axis to a logarithmic scale, right-click on any graph and then select "Use Logarithmic X Axis" (1) in the context menu. If multiple graphs are displayed as shown in the figure below, all X-axes are switched to a logarithmic scale. To restore the linear scale, right-click again on any graph and then select "Use Linear X Axis" in the context menu.

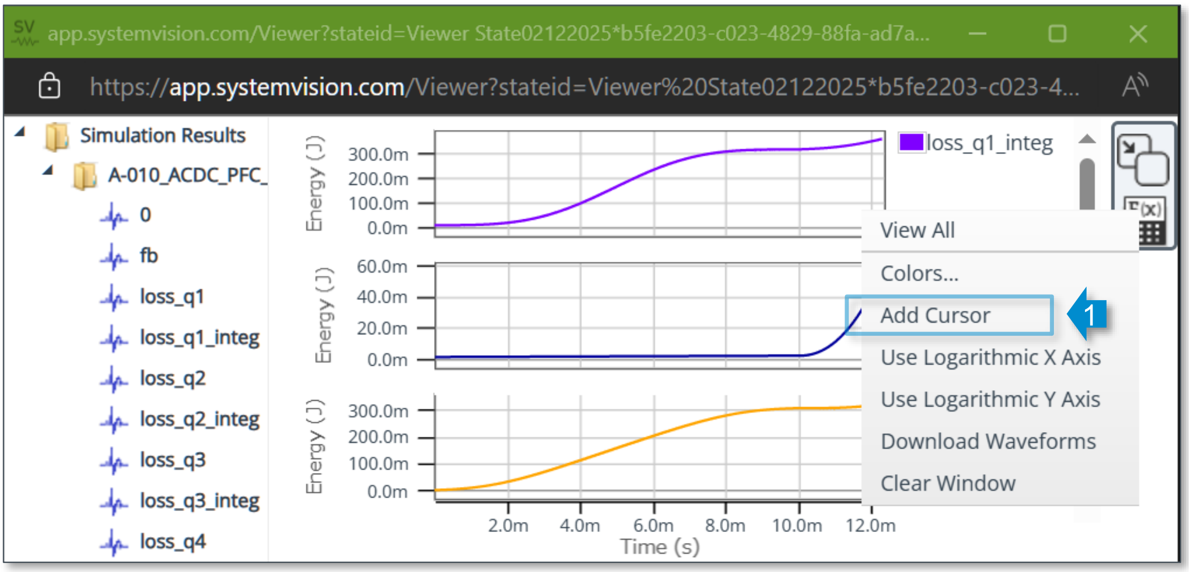
To switch the Y-axis of a graph to a logarithmic scale, right-click on the inside of the graph where no waveform exists and then select "Use Logarithmic Y Axis" (2) in the context menu. For the Y-axis, a logarithmic or linear scale can be selected for each graph. To restore the linear scale, right-click on the inside of the graph where no waveform exists and then select "Use Linear Y Axis" in the context menu.



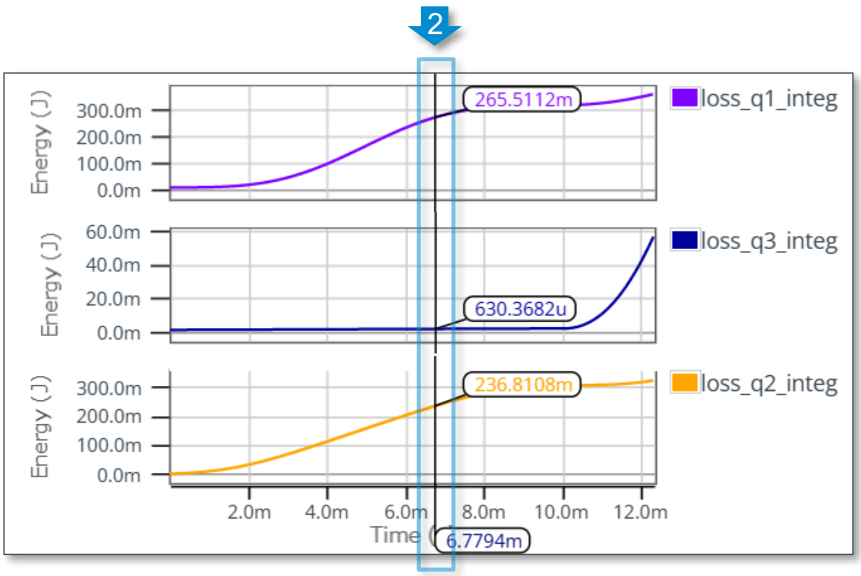
Waveform Cursors

You can display a cursor to read numerical values from waveforms.

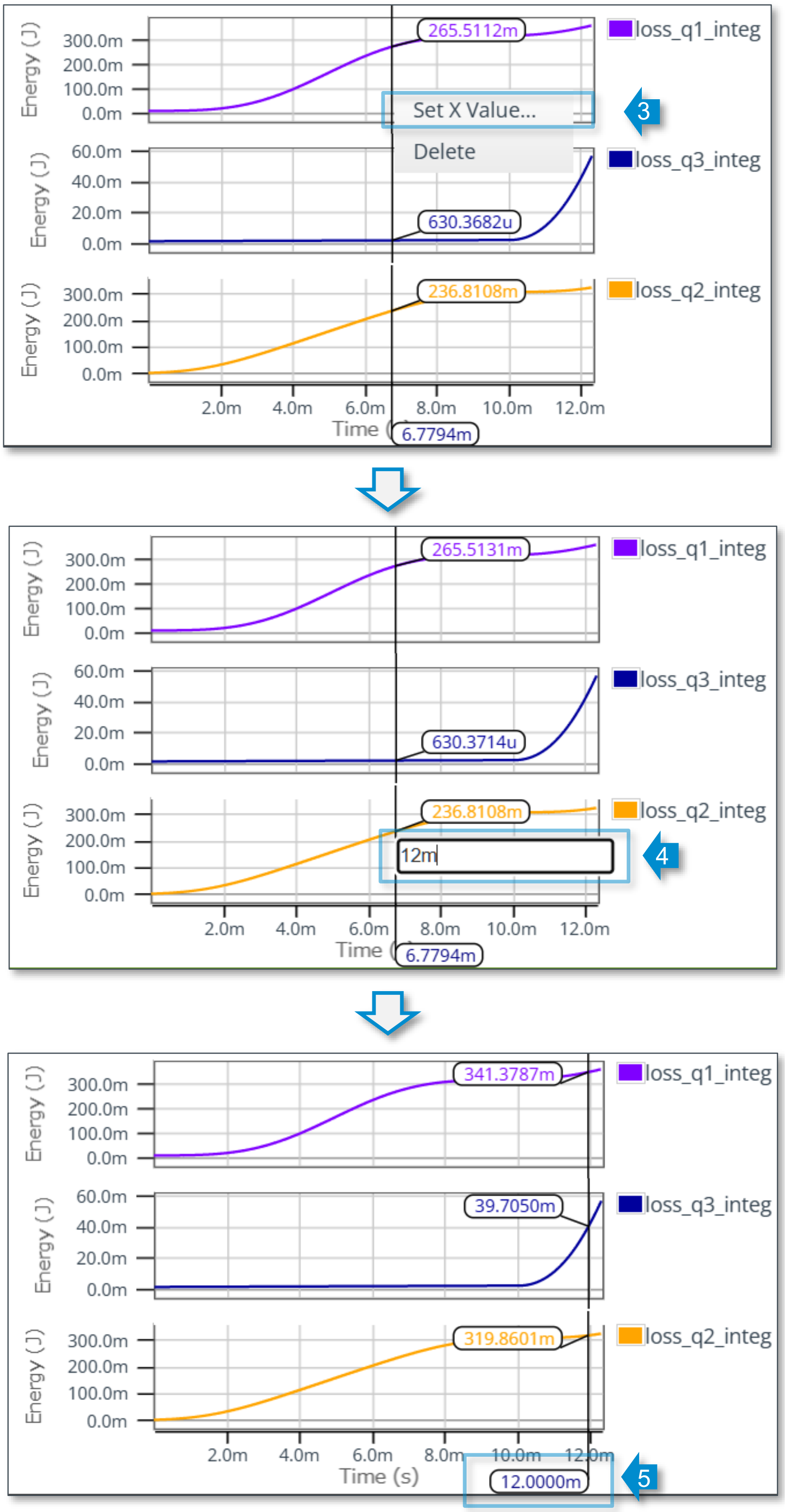
Right-click on the graph and then select "Add Cursor" (1) from the context menu. You can display multiple cursors by selecting "Add Cursor" again.



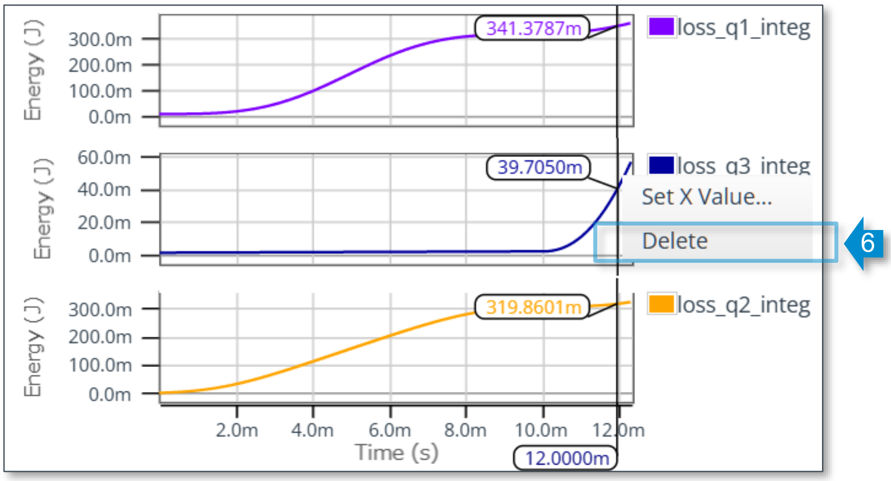
A cursor is added, displaying the numerical values on the X- and Y-axes. You can drag and move the cursor to the left and right (2).



To move the cursor to a specified X-axis value, right-click on the cursor and then select "Set X Value..." (3) in the context menu. When the input field appears, enter a numerical value on the X-axis (4). The cursor is moved to the desired position (5).

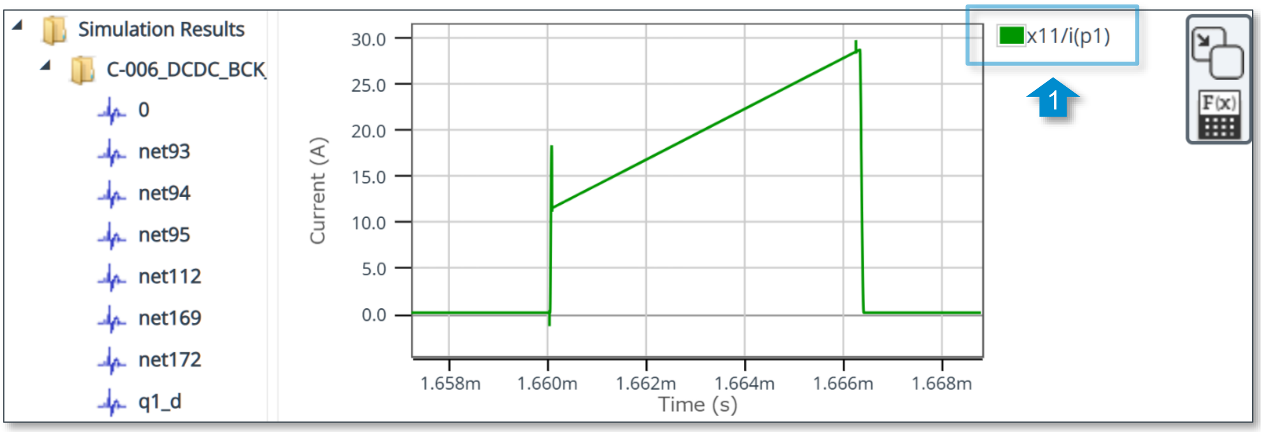


To delete an unnecessary cursor, right-click on the cursor and then select “Delete” (6) in the context menu. Note that you may not be able to delete the cursor if it is displayed initially for a circuit diagram provided by ROHM.

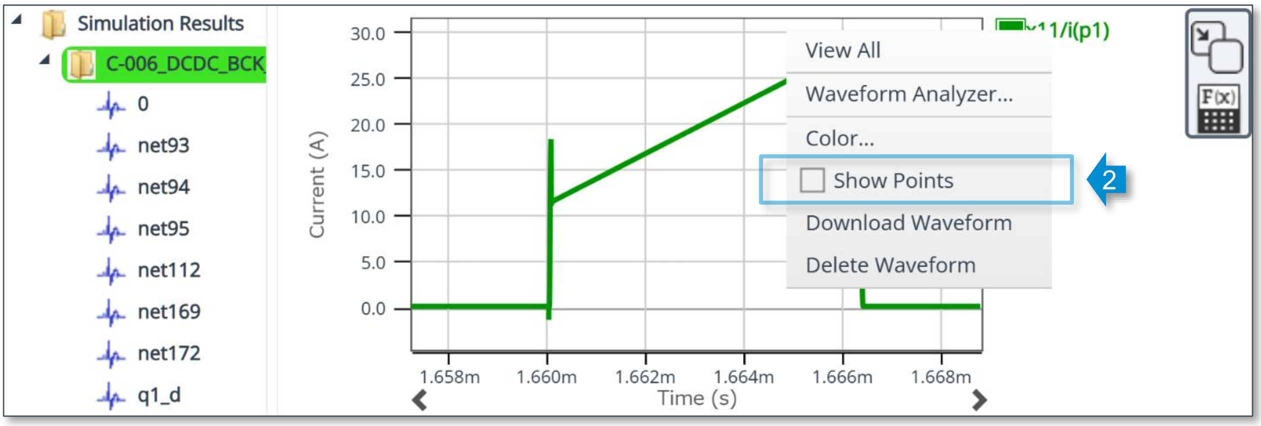


Show the calculated points

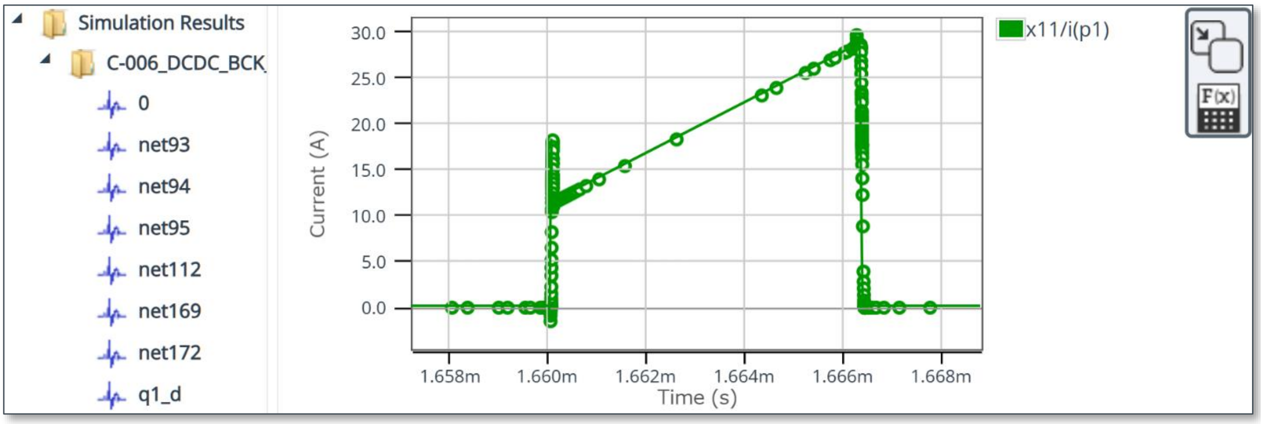
You can display the calculated points on the waveform to check whether the calculation is performed correctly for a sudden change in values. Right-click on the signal name (1) on the upper right of the graph.



Select "Show Points" (2) in the context menu.



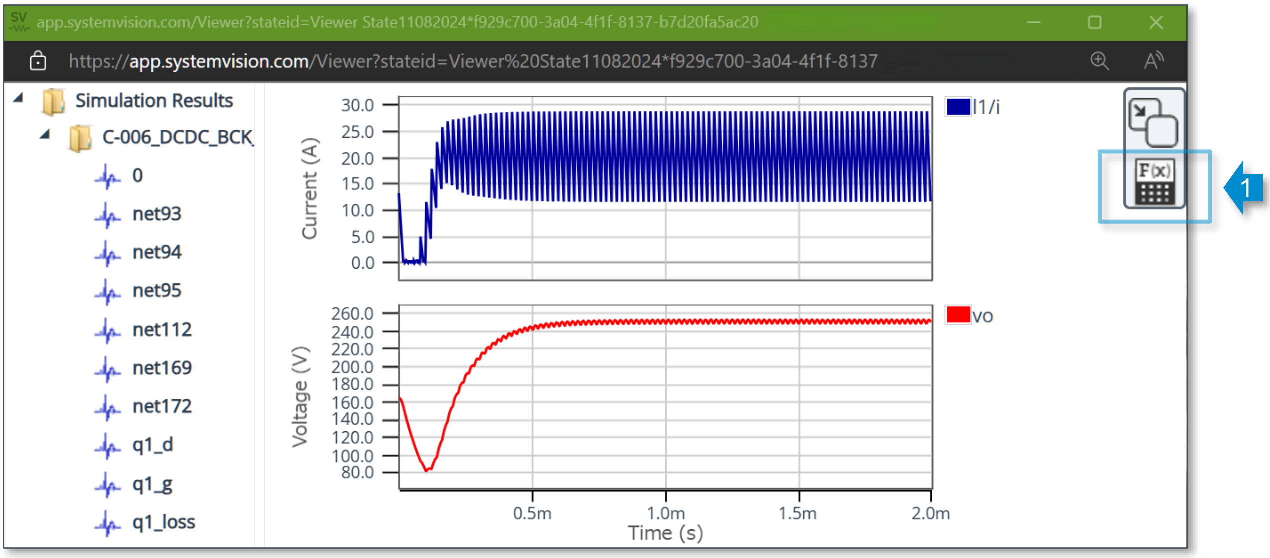
The calculated points are displayed on the waveform.



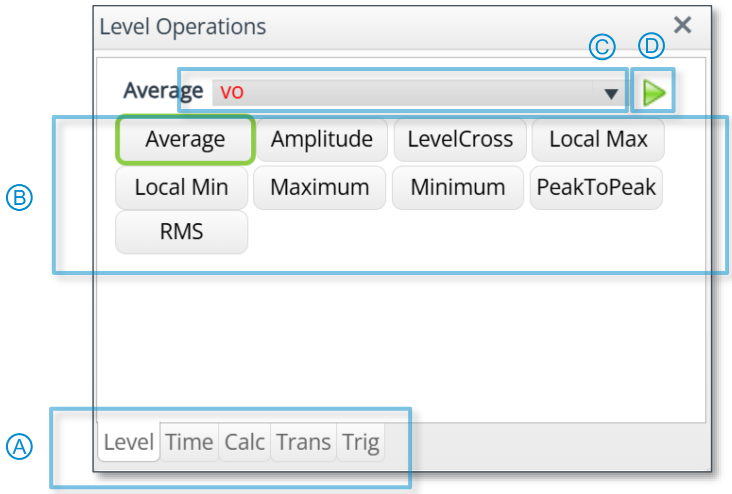
Waveform Analyzer

This tool is equipped with various waveform measurement and calculation functions.

Click the calculator icon (1) on the upper right to open the tool dialog.



Elements of the Waveform Analyzer



Ⓐ Function group tab

The measurement/calculation functions are categorized into groups.

Ⓑ Function buttons

The measurement/calculation functions are lined up. Select the button of a function you wish to execute.

Ⓒ Node name selection

In the pull-down list, select the name of a node for which you wish to perform measurement/calculation.

Ⓓ Apply button

Applies the waveform measurement/calculation.

Function Description

See below for features available in Waveform Analyzer.

Level

Function	Description
Average	The average level computed by integrating the waveform and dividing by the duration of the waveform.
Amplitude	The difference between the Topline and Baseline levels.
Level Cross	The X value(s) where the waveform is equal to the specified level.
Local Max	The local maximum points on the waveform.
Local Min	The local minimum points on the waveform.
Maximum	The maximum value of the waveform.
Minimum	The minimum value of the waveform.
Peak To Peak	The difference between the Maximum and the Minimum values of a waveform.
RMS	The root-mean-square value of the waveform.

Time (Time-Domain)

Function	Description
Duty Cycle	The ratio of the “high” portion of a cycle in the waveform to the length of the Period.
Fall time	The X difference between the upper and lower levels of the waveform.
Frequency	The reciprocal of the Period, where the period is calculated as the X difference between two consecutive edges of the same polarity.
Overshoot	The difference between the maximum value of a pulse and the topline level.
Period	The X difference between two consecutive edges of the same polarity on the waveform.
Pulse Width	The width of a pulse that starts with a rising edge and ends with the next falling edge (a positive pulse) or that starts with a falling edge and ends with the next rising edge (a negative pulse).
Risetime	The x difference between the lower and upper levels of the waveform.
Settle Time	The time taken for the waveform to settle to within a tolerance band of the topline level (in the case of a rising edge) or the baseline level (in case of a falling edge).
Slew Rate	The ratio of the Y difference between the upper and lower levels to the risetime (in the case of a rising edge) or the fall time (in the case of a falling edge).
Slope	The slope of a waveform at the specified X value.
Undershoot	The difference between the baseline level and the minimum value of a negative pulse.

Calc (Calculator)

Function	Description
÷ × − +	Performs the indicated arithmetic operation (Divide, Multiply, Subtract, Add)
0 ～ 9	Inserts an integer.
.	Inserts a decimal point.
←	Back space.
+/−	Changes a positive number to negative or a negative number to positive.
E	Enter numbers using exponential notation.
π	Inserts an approximate value of pi (3.141592653589793).

Trans (Transformations)

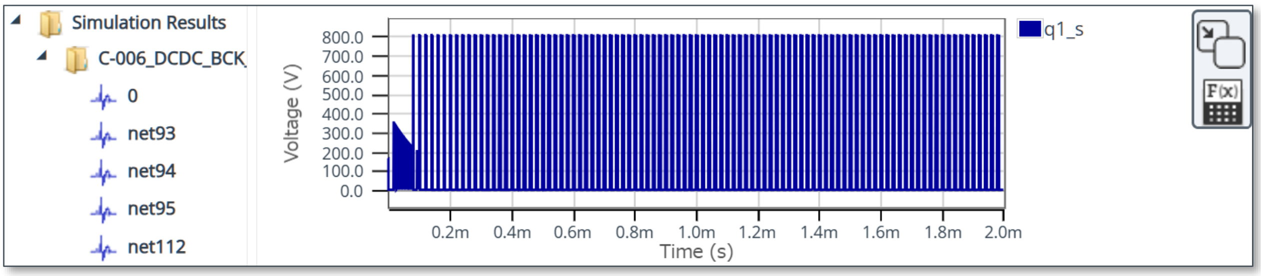
Function	Description
dy/dt	Returns the derivative of a waveform.
$\int$	Returns the anti-derivative of a waveform.
e^x	Returns the exponential function of a waveform.
$\ln$	Returns the natural logarithm of a waveform.
x^2	Returns the squared value of a waveform.
$\sqrt{x}$	Returns the square root of a waveform.
10^x	Returns 10 to the xth power of a waveform.
$\log_{10}$	Returns the base 10 logarithm of a waveform.
$1/x$	Returns the reciprocal value of a waveform.
x^y	Returns x to the yth power (x**y), where x and y can be either a waveform.
$\|x\|$	Returns the absolute value of waveform, the absolute value of each data point is computed.
WinAve	Compute the average over the waveform using a sliding window.
DtoA	Transform Bit, Boolean or std_logic waveform to analog.
AtoD	Transform Analog waveform to std_logic.
XLimit	Limit X range of a waveform.
FFT	Compute the discrete Fourier transform of the waveform.
YvsX	Y versus X of two waveforms.

Trig (Trigonometric)

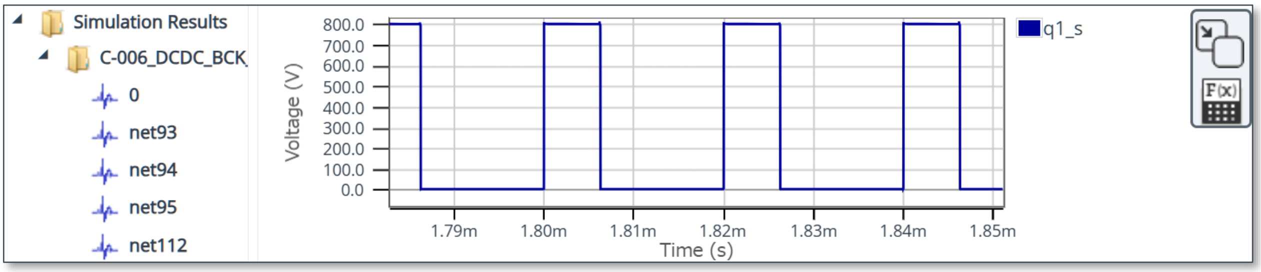
Function	Description
sin	Sine
asin	Arc sine
cos	Cosine
acos	Arc cosine
tan	Tangent
atan	Arc tangent
sinh	Hyperbolic sine
asinh	Hyperbolic arc sine
cosh	Hyperbolic cosine
acosh	Hyperbolic arc cosine
tanh	Hyperbolic tangent
atanh	Hyperbolic arc tangent

Usage example 1: Measure the frequency of a switching waveform.

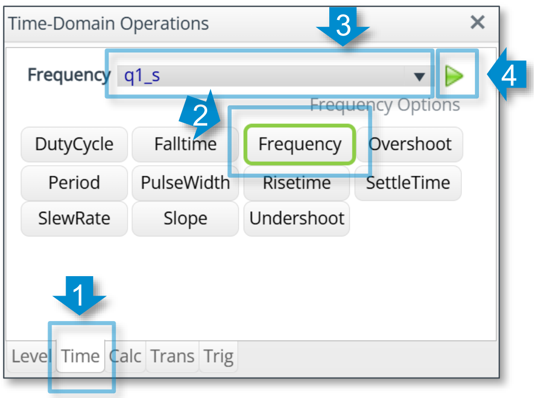
1. The simulated waveform is shown below. If the measurement is applied in this state, the Waveform Analyzer automatically selects and measures a certain section. Due to this, an appropriate result may not be obtained. Therefore, zoom in on a section of the waveform you wish to measure before applying the measurement.



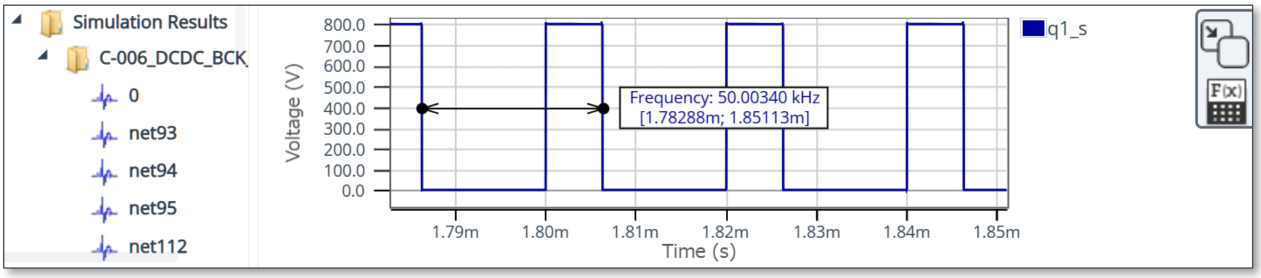
2. The waveform enlarged with the mouse is shown in the figure below.



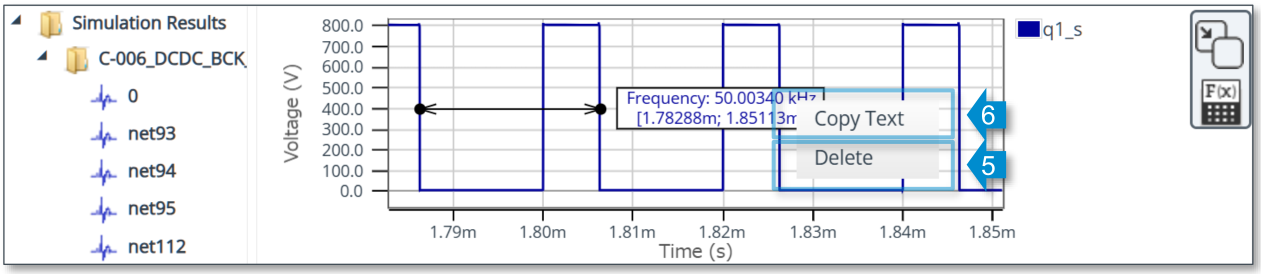
3. In the Waveform Analyzer dialog, select the “Time” tab (1) and then select the “Frequency” button (2). If multiple signals are displayed, select the name of the signal to be measured (3). Finally, click the “Apply” button (4).



4. The measurement result is displayed.

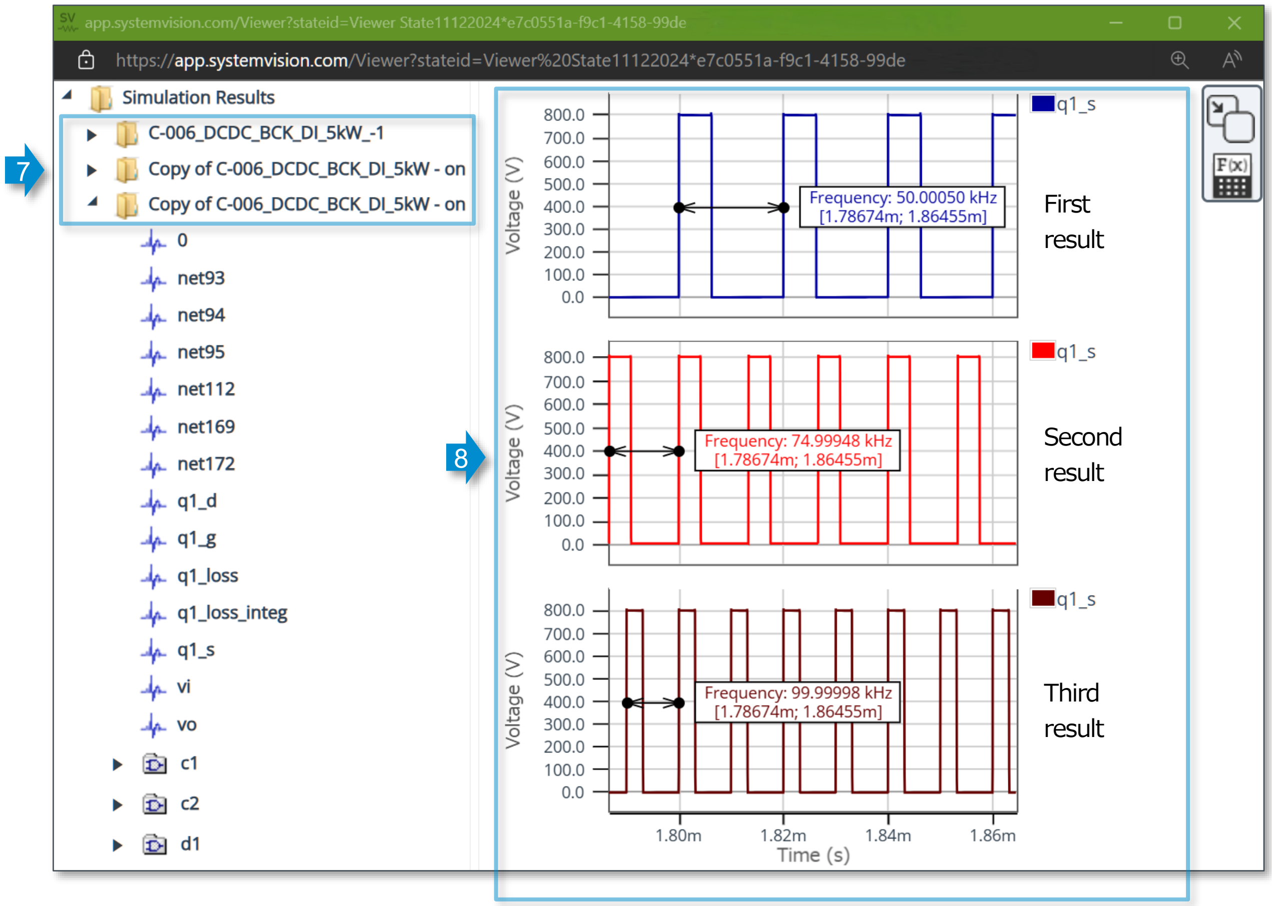


5. To delete the measurement result, right-click on the result display and then select "Delete" (5) in the context menu. In addition, selecting "Copy Text" (6) copies the measurement result to the Windows clipboard.



Usage example 2: Make comparisons with previous simulation results.

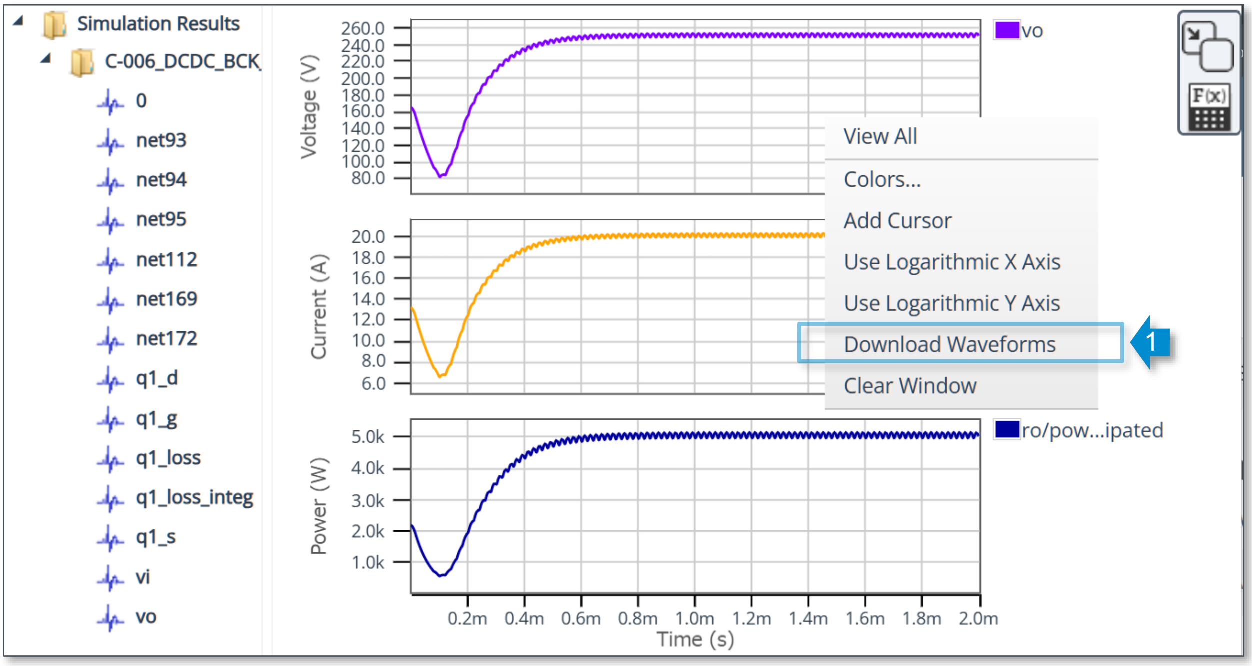
Every time a simulation is executed, a folder is added to the simulation result display area in the left column (7). In this example, there are three folders because the simulation has been executed three times. Selecting the same signal name in the folders allows you to compare the waveform changes (8).



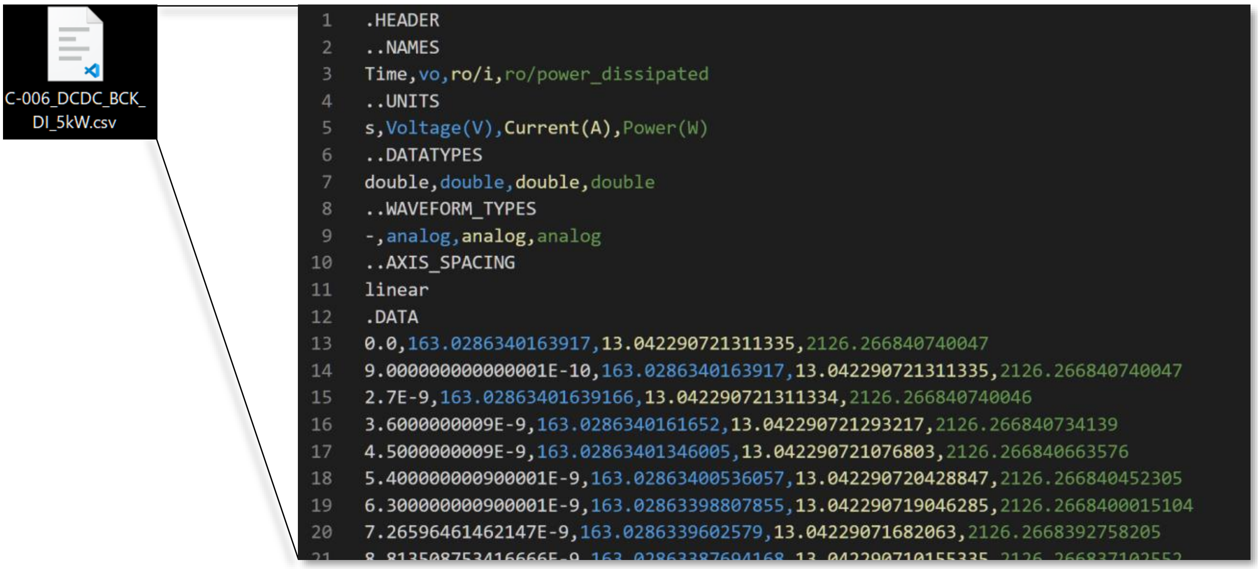
Download Waveforms

You can download the waveform data as a CSV file and utilize it with other software.

Right-click on the Waveform Viewer and then select "Download Waveforms" (1) in the context menu.



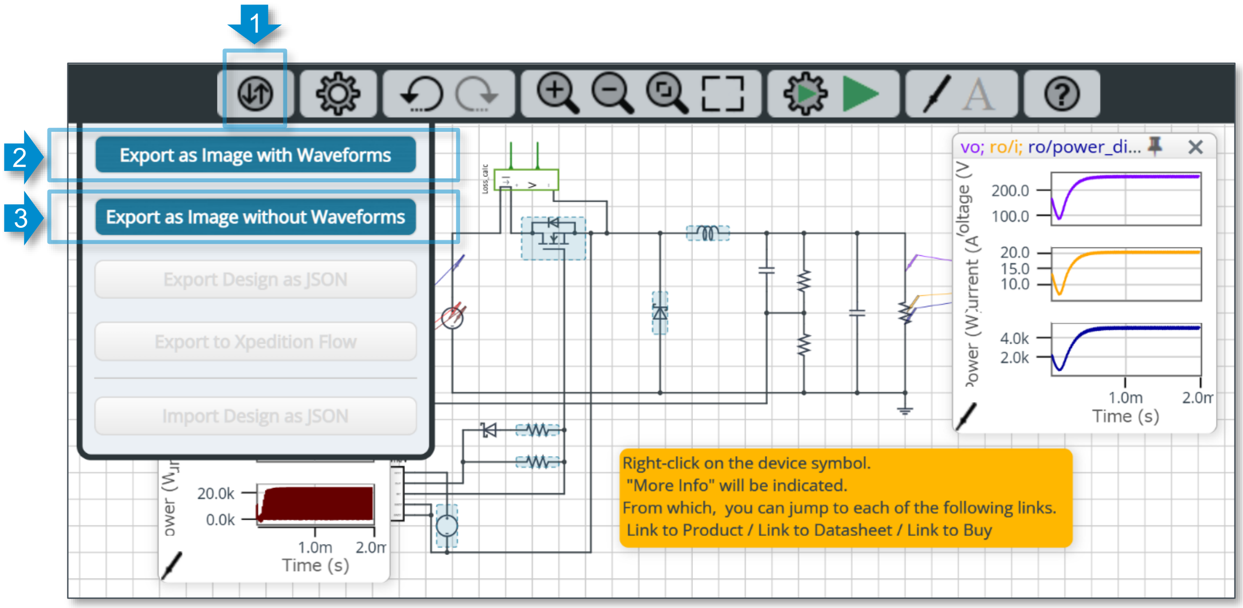
Wait for the data to be processed on the server and the download to complete. Note that it may take several tens of seconds to several minutes if there are many data points.



12. Exporting Circuit Data

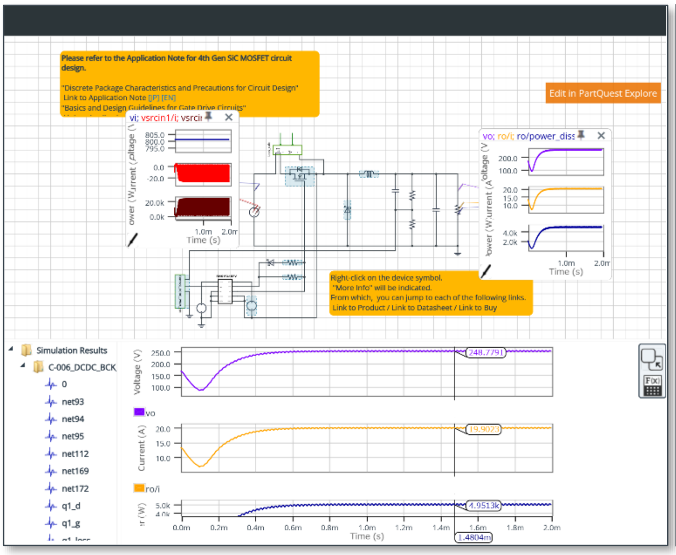
12-1. Image Files

The circuit diagram and waveforms can be downloaded as image data (PNG file). Click the "Export/Import" icon (1) on the tool bar and then select "Export as Image with Waveforms" (2) in the drop-down menu to export the circuit diagram and waveforms and download them as image data. To include the waveforms of the Waveform Viewer in the image, keep the Waveform Viewer at the bottom of the screen open. Selecting "Export as Image without Waveforms" (3) exports the image data of the circuit diagram for download with the waveforms hidden.

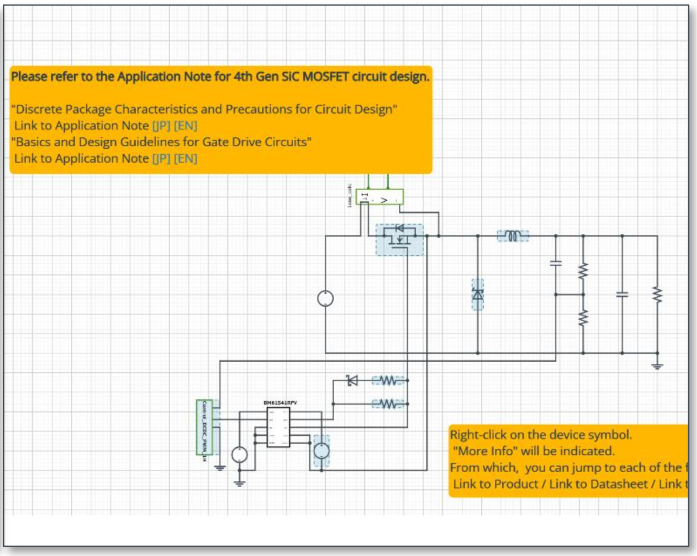


Example of downloading image data: The image is similar to a screenshot.

Circuit diagram with waveforms

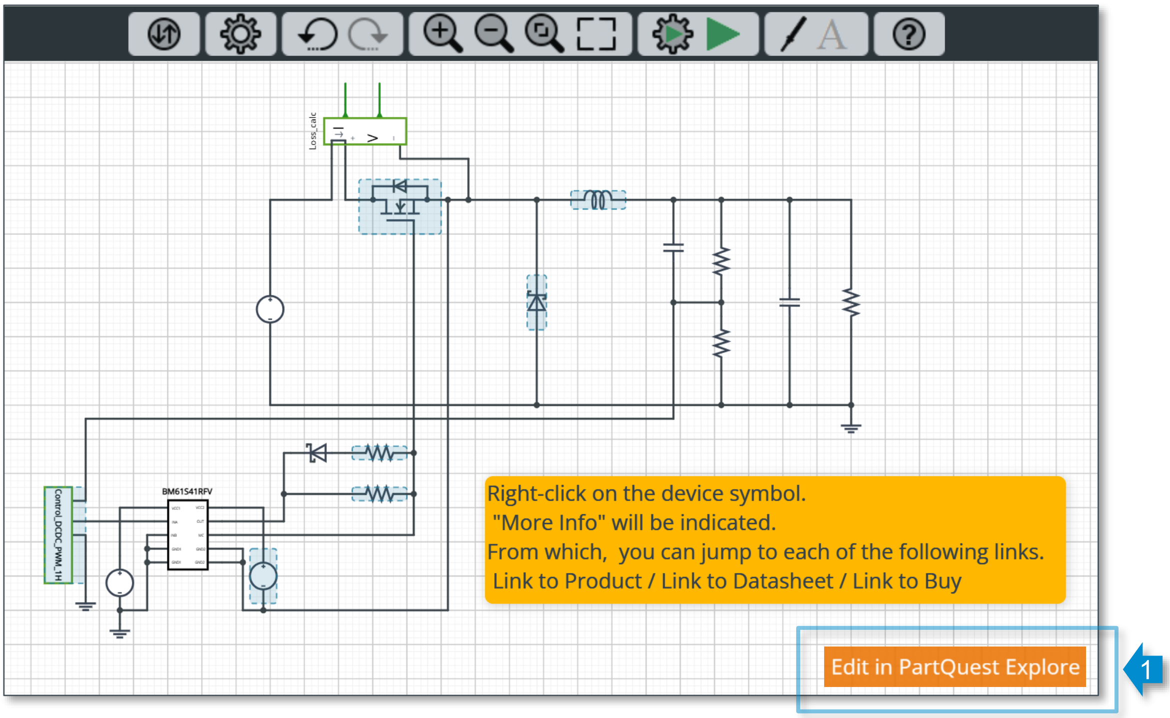


Circuit diagram without waveforms

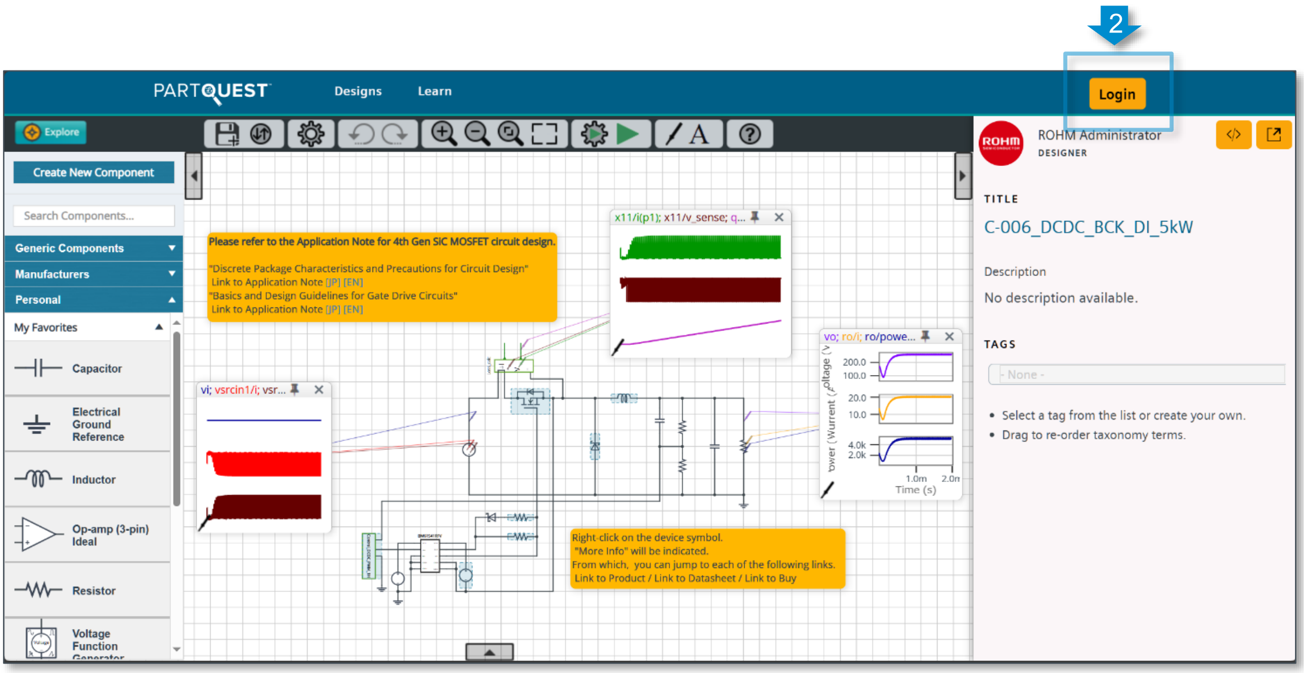


12-2. Simulation Data

Exporting circuits from the ROHM Solution Simulator to PartQuest Explore allows you to edit and save circuit diagrams. Click the “Edit in PartQuest Explore” button (1) on the lower right of the circuit diagram to export the circuit diagram to PartQuest Explore.



The circuit diagram is opened in PartQuest Explore. Log in on the upper right of the screen (2) to enable various functions. The following steps require a SIEMENS account. If you do not have one, please register.



12-3. Differences between ROHM Solution Simulator and PartQuest™ Explore

The table below shows a comparison of the ROHM Solution Simulator and PartQuest™ Explore.

<table>
<tr><th>Item</th><th>ROHM Solution Simulator</th><th colspan="2">PartQuest™ Explore</th></tr>
<tr><td>ROHM application circuits</td><td>Provided</td><td colspan="2">Not provided</td></tr>
<tr><td>User registration</td><td>MyROHM account</td><td colspan="2">SIEMENS account</td></tr>
<tr><td>Operation environment</td><td>Web browser</td><td colspan="2">Web browser</td></tr>
<tr><td>Internet environment</td><td>Necessary</td><td colspan="2">Necessary</td></tr>
<tr><td>URL</td><td>https://www.rohm.com/solution-simulator</td><td colspan="2">https://explore.partquest.com/</td></tr>
<tr><td>Software installation</td><td>Not necessary</td><td colspan="2">Not necessary</td></tr>
<tr><td>Simulator engine</td><td>PartQuest™ Explore</td><td colspan="2">PartQuest™ Explore</td></tr>
<tr><td>Simulation speed</td><td>Depends on traffic of cloud server</td><td colspan="2">Depends on traffic of cloud server</td></tr>
<tr><td>Destination of circuit diagrams to save</td><td>Cannot be saved</td><td colspan="2">SIEMENS cloud</td></tr>
<tr><td>Number of circuits to be saved</td><td>–</td><td colspan="2">No limit</td></tr>
<tr><td>Number of elements</td><td>–</td><td colspan="2">No limit</td></tr>
<tr><td rowspan="2">Secrecy of circuit diagrams</td><td rowspan="2">–</td><td>Free plan</td><td>Subscription</td></tr>
<tr><td>Public</td><td>Private</td></tr>
<tr><td rowspan="3">Execution time</td><td rowspan="3">Up to 1 hour each
(not the TSTOP setting time for transient analysis)</td><td>Free plan</td><td>Subscription</td></tr>
<tr><td>Circuit provided by ROHM
Up to 1 hour each</td><td>Circuit provided by ROHM
Up to 1 hour each</td></tr>
<tr><td>Newly created
Up to 1 minute each</td><td>Newly created
Up to 1 hour each</td></tr>
<tr><td>Editing of circuit diagrams</td><td>Not possible</td><td colspan="2">Possible</td></tr>
<tr><td>Modification of component properties</td><td>Partially possible</td><td colspan="2">Possible</td></tr>
<tr><td>Addition of components</td><td>Not possible</td><td colspan="2">Possible</td></tr>
</table>

As of February 2025

PartQuest™, HyperLynx™, Xpedition™ is a registered trademark of Siemens Digital Industries Software, Inc.

Notice

1) The information contained in this document is intended to introduce ROHM Group (hereafter referred to asROHM) products. When using ROHM products, please verify the latest specifications or datasheets before use.

2) ROHM products are designed and manufactured for use in general electronic equipment and applications (such as Audio Visual equipment, Office Automation equipment, telecommunication equipment, home appliances, amusement devices, etc.) or specified in the datasheets. Therefore, please contact the ROHM sales representative before using ROHM products in equipment or devices requiring extremely high reliability and whose failure or malfunction may cause danger or injury to human life or body or other serious damage (such as medical equipment, transportation, traffic, aircraft, spacecraft, nuclear power controllers, fuel control, automotive equipment including car accessories, etc. hereafter referred to as Specific Applications). Unless otherwise agreed in writing by ROHM in advance, ROHM shall not be in any way responsible or liable for any damages, expenses, or losses incurred by you or third parties arising from the use of ROHM Products for Specific Applications.

3) Electronic components, including semiconductors, can fail or malfunction at a certain rate. Please be sure to implement, at your own responsibilities, adequate safety measures including but not limited to fail-safe design against physical injury, and damage to any property, which a failure or malfunction of products may cause.

4) The information contained in this document, including application circuit examples and their constants, is intended to explain the standard operation and usage of ROHM products, and is not intended to guarantee, either explicitly or implicitly, the operation of the product in the actual equipment it will be used. As a result, you are solely responsible for it, and you must exercise your own independent verification and judgment in the use of such information contained in this document. ROHM shall not be in any way responsible or liable for any damages, expenses, or losses incurred by you or third parties arising from the use of such information.

5) When exporting ROHM products or technologies described in this document to other countries, you must abide by the procedures and provisions stipulated in all applicable export laws and regulations, such as the Foreign Exchange and Foreign Trade Act and the US Export Administration Regulations, and follow the necessary procedures in accordance with these provisions.

6) The technical information and data described in this document, including typical application circuits, are examples only and are not intended to guarantee to be free from infringement of third parties intellectual property or other rights. ROHM does not grant any license, express or implied, to implement, use, or exploit any intellectual property or other rights owned or controlled by ROHM or any third parties with respect to the information contained herein.

7) No part of this document may be reprinted or reproduced in any form by any means without the prior written consent of ROHM.

8) All information contained in this document is current as of the date of publication and subject to change without notice. Before purchasing or using ROHM products, please confirm the latest information with the ROHM sales representative.

9) ROHM does not warrant that the information contained herein is error-free. ROHM shall not be in any way responsible or liable for any damages, expenses, or losses incurred by you or third parties resulting from errors contained in this document.

ROHM SEMICONDUCTOR

Thank you for your accessing to ROHM product informations.
More detail product informations and catalogs are available, please contact us.

ROHM Customer Support System

https://www.rohm.com/contactus