UM1874
User Manual

TwisterSIM

Introduction

TwisterSIM is a unique **Electro-Thermal simulator** that helps shorten the design solution cycle by enabling, in a few clicks, complex engineering evaluations with accurate simulations like load-compatibility, wiring harness optimization, fault condition impact analysis, **diagnostic behavior analysis and Dynamic Thermal performance**.

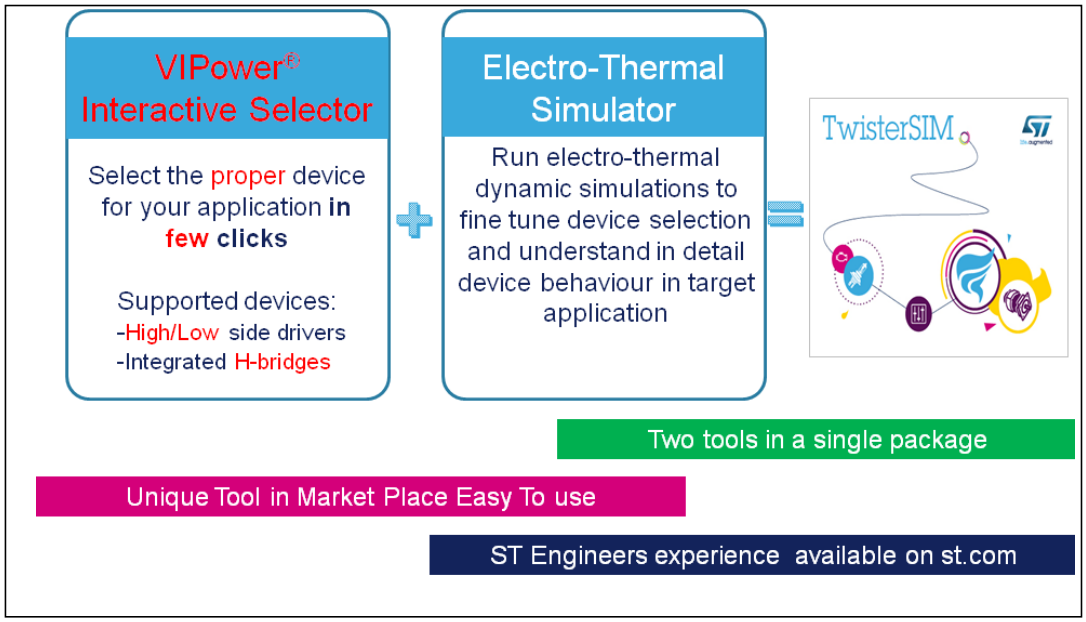
A built-in Interactive Selector provides a short list of suitable devices based on first level system requirements. It assists in detailing the specific system configuration with layout, load and driving profile customization to build an accurate model of the final application.

Simulation results, including junction and case thermal profiles, load current and diagnostic behavior are shown on dedicated scopes views or exported in a number of different commonly used formats.

TwisterSIM supports a large selection of **Low/High-side driver/switches and H-bridges for Motor Control**.

A trial version, to explore the main functionalities of TwisterSIM, is available for download at *www.st.com/twistersim* and it also contains, in the main toolbar, the instructions to request an activation code (free of charge but subject to STMicroelectronics approval) for the full version.

An on-line forum provides additional support to TwisterSIM users. Access to the forum is available at *www.st.com/twistersim-forum*.



Contents

1 **Overview** 6

2 **Electro thermal simulator** 8

2.1 Device selection and parameters setting 8

2.1.1 General device parameters 8

2.1.2 H-Bridge Device parameters 9

2.1.3 24 V Device parameters 9

2.1.4 M07 Device parameters 9

2.1.5 Multichannel Options (2 or 4 channels devices) 9

2.2 Nominal voltage selection and setting 10

2.2.1 DC voltage setting 10

2.2.2 Voltage profile setting 10

2.3 Line and sense resistances 12

2.3.1 Line-in resistor 12

2.3.2 Line-out resistors 12

2.3.3 Sense resistors 13

2.4 Control source 14

2.4.1 Control source type 15

2.4.2 H-Bridge input setting 16

2.5 Load selection 17

2.5.1 Resistor as load 18

2.5.2 RC parallel as load 19

2.5.3 RL series as load 20

2.5.4 Current, RL and device power loss profile as load 20

2.5.5 PMDC machine as load 22

2.5.6 Lamp as load 24

2.6 Thermal electrical simulation setup 25

2.7 Graphical simulation results 27

2.8 Monte Carlo analysis 30

2.8.1 Monte Carlo parameters setting 31

2.8.2 Graphical Simulation Results (MC analysis) 33

3 **VIPower interactive selector overview** 35

3.1 Select application line and nominal voltage levels 35
3.2 Select device topology 35
3.3 Select number of channels 36
3.4 Select load type 36
3.5 Select source type 38
3.6 Set ambient & initial device temperature 38
3.7 Select PCB type 38
3.8 Device list 39

4 Revision history 40

List of tables

Table 1. Document revision history. 40

List of figures

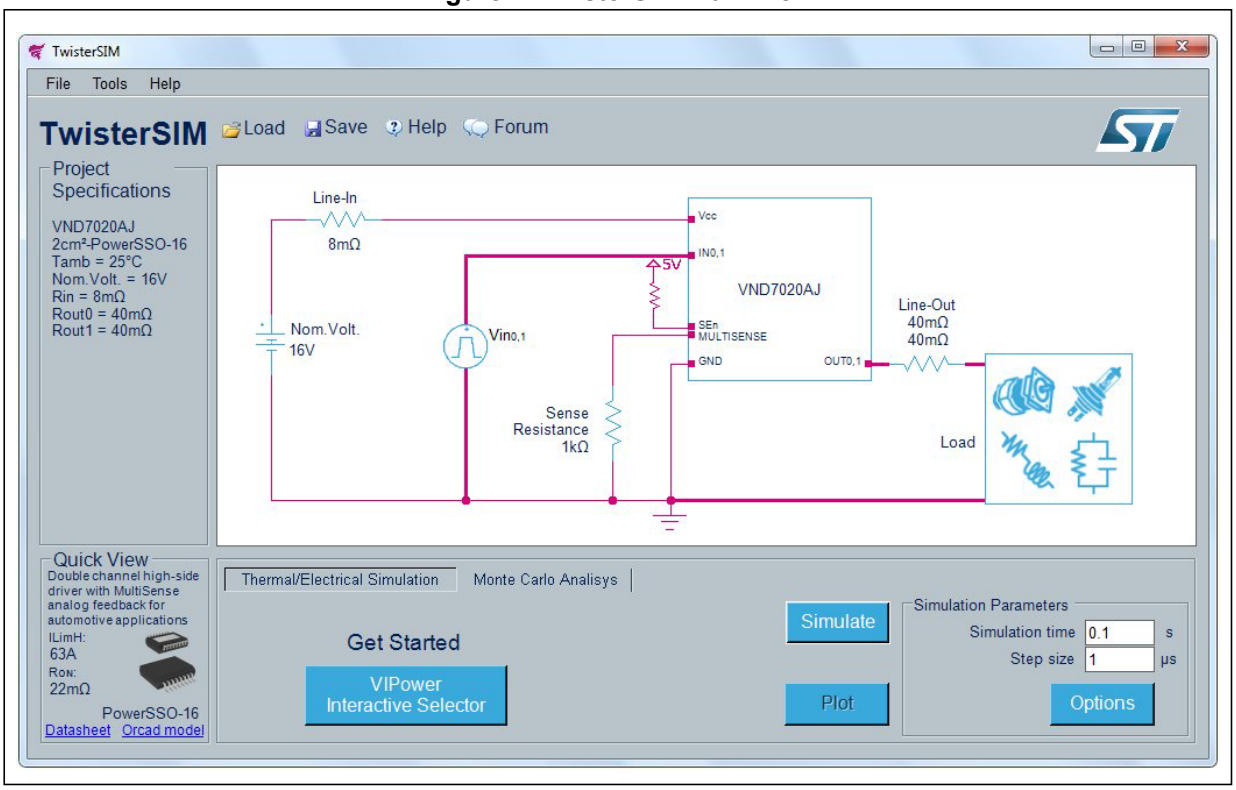
Figure 1. TwisterSIM main view . . . 6
Figure 2. Device selection and parameters setting . . . 8
Figure 3. DC voltage setting . . . 10
Figure 4. Voltage profile setting . . . 11
Figure 5. Line-in resistor . . . 12
Figure 6. Line-out resistors . . . 13
Figure 7. Sense resistors . . . 14
Figure 8. Control source type . . . 15
Figure 9. PWM profile selection . . . 16
Figure 10. H-Bridge input setting . . . 17
Figure 11. Load selection . . . 18
Figure 12. Resistor as load . . . 19
Figure 13. RC parallel as load . . . 19
Figure 14. RL series as load . . . 20
Figure 15. Output current . . . 21
Figure 16. RL load . . . 21
Figure 17. Device power loss . . . 22
Figure 18. PMDC machine as load . . . 23
Figure 19. Lamp as load . . . 24
Figure 20. Lamp harness resistors . . . 25
Figure 21. Thermal electrical simulation setup . . . 26
Figure 22. Simulation options . . . 26
Figure 23. Simulation progress bar . . . 27
Figure 24. Graphical simulation results . . . 28
Figure 25. Create a new function to plot . . . 29
Figure 26. Monte Carlo analysis simulation setup . . . 30
Figure 27. Save Monte Carlo analysis simulation data . . . 30
Figure 28. Monte Carlo analysis wait for simulation . . . 31
Figure 29. Monte Carlo parameters setting . . . 31
Figure 30. Gaussian distribution diagram . . . 32
Figure 31. Graphical simulation results (MC analysis) . . . 33
Figure 32. VIPower interactive selector . . . 35
Figure 33. Select application line and nominal voltage levels . . . 35
Figure 34. Select device topology . . . 35
Figure 35. Select number of channels . . . 36
Figure 36. Select load type . . . 36
Figure 37. Resistor as load . . . 36
Figure 38. Inductor as load . . . 36
Figure 39. DC Current as load . . . 37
Figure 40. Lamp (by current) as load . . . 37
Figure 41. Lamp (by Power) as load . . . 37
Figure 42. PMDC motor as load . . . 37
Figure 43. Select source type . . . 38
Figure 44. Set ambient & initial device temperature . . . 38
Figure 45. Select PCB type . . . 38
Figure 46. Device list . . . 39

1 Overview

Main start window (*Figure 1*) functionalities:

1. System Layout: it shows a typical control schematic, including a specific device (selectable from a list), supply voltage, line harnesses, common loads and a configurable control input to fit the simulation conditions to a specific architecture
2. Project Specifications: on the left side, the main project information is summarized
3. Simulation pane: on bottom-right part, it allows to customize simulation parameters and open a plot window
4. VIPower Interactive Selector: built-in "Get Started" utility to facilitate device identification based on first level system requirements
5. Quick View: on the bottom side the main electrical data and package info of the device are outlined. The link View Datasheet allows downloading the datasheet.

Figure 1. TwisterSIM main view



On menu items or toolbar:

- Click "File" and select "Load System Layout & Data" or click button, to load a complete system layout and simulation data, previously saved
- Click "File" and select "Save System Layout & Data" or click button, to save a complete system layout and simulation data
- Click "Tools" and select "VIPower Interactive Selector" or click "Get Started" button, to start VIPower Interactive Selector tool
- Click “Help” to view this guide and the disclaimer.
- Click button to visit TwisterSIM forum page.

On system layout:

- Click on the components icons to configure and change their values.

On simulation pane:

- Configure simulation parameters (*Section 2.6: Thermal electrical simulation setup*) then click "Simulate" button to start simulation.

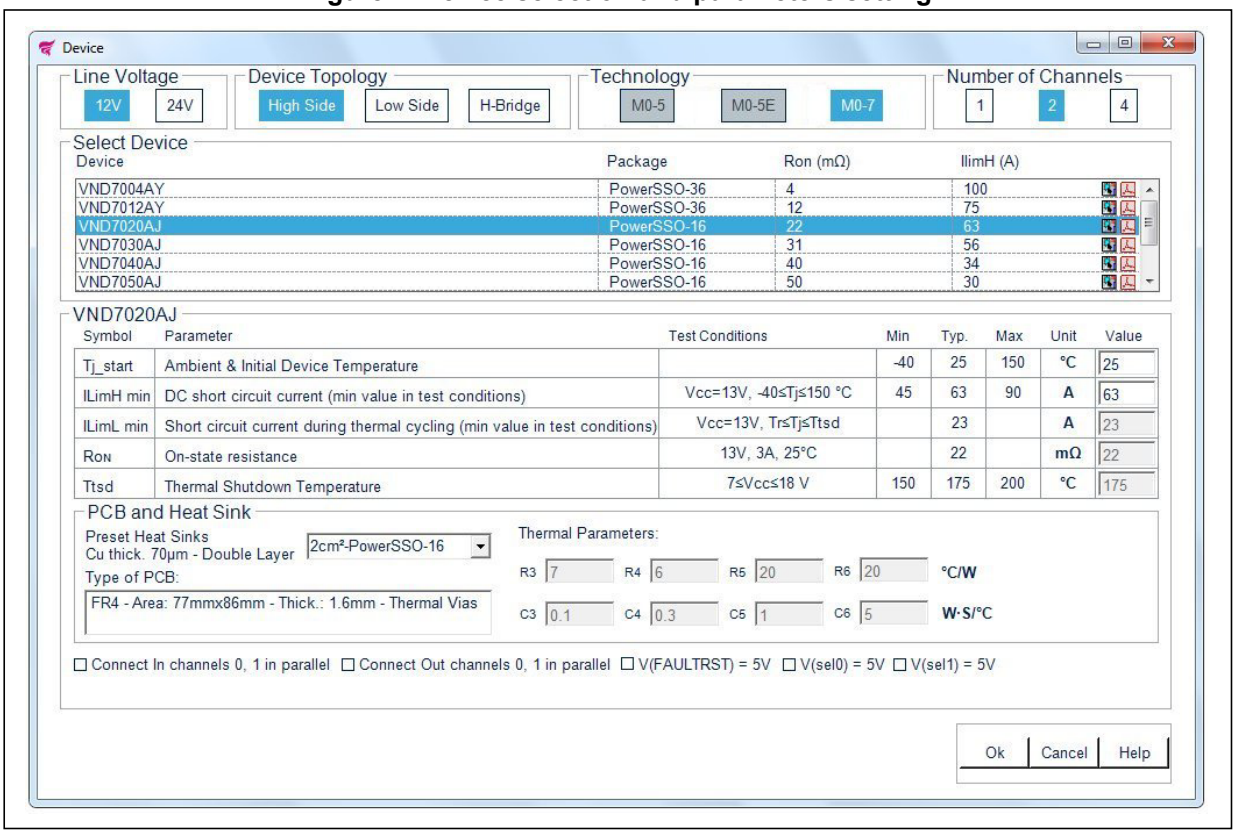
A progress bar indicates the state of the simulation. A plotting window automatically opens when the simulation ends.

2 Electro thermal simulator

2.1 Device selection and parameters setting

Click on the Device Icon in the main window, a new frame opens (*Figure 2*) where a specific device can be selected and the related parameters set.

Figure 2. Device selection and parameters setting



2.1.1 General device parameters

Typical values are preselected for each device. All parameters with a white background may be changed.
Change them manually by inserting a specific value in the numeric box.

The range of values allowed depends on the specific device selected:

- Ambient and Junction temperature at the beginning of the simulation. Accepted values are between -40 °C and 150 °C for all devices
- ILimH min (A): DC short circuit current (minimum value in test conditions for Tj < Ttsd).
- ILimL min (A): short circuit current during thermal cycling (minimum value in test conditions for Tj > Ttsd).
- RdsOn (mΩ): output drain-source on state resistance at 25 °C.
- Ttsd (°C): Thermal Shutdown Temperature. Accepted values are between 150°C and 200°C for all devices.
- Cx (°C/W) and Rx (W · S/°C): Thermal Fitting model parameters (see component datasheet for further information).

In PCB pop-up Menu choose from a list of preset Heat Sink (with preset values of Cx and Rx, according to the relevant datasheet) or set "Custom PCB" to manually insert Cx and Rx values.
Values outside the boundaries are automatically restored to the max/min allowed value.

2.1.2 H-Bridge Device parameters

- ILimH min (A): High-Side Current Limitation (minimum value in test conditions).
- ISD_LS (A): Low-Side Shutdown Current.
- Ron HS/LS (mΩ): high/low side drain-source on state resistances at 25 °C.

H-Bridge Device selection is not available if Power Profile is selected.
H-Bridge Device selection disables Power Profile, Lamp Selection and Reverse Diode selection in parallel with RL load.

2.1.3 24 V Device parameters

- V(FR_Stby) = 5: Check to connect Fault reset standby pin at 5 V.

2.1.4 M07 Device parameters

- V(FAULTRST) = 5: Check to connect Fault reset pin at 5 V
- V(sel0) = 5: Check to connect sel0 pin at 5 V
- V(sel1) = 5: Check to connect sel1 pin at 5 V.

2.1.5 Multichannel Options (2 or 4 channels devices)

- Check "connect In channels in parallel" to connect the Inputs of the channels in parallel. Relevant changes are reflected in the "source section"
- Check "connect Out channels in parallel" to connect the Outputs of the channels in parallel. Relevant changes are reflected in the "load section"
- Click **OK** to accept changes
- Click **Cancel** to restore previous device and parameters
- Click **Help** to read the help file on this topic.

Selected Device name and PCB thermal parameters are reported in the main window. Refer to the relevant datasheet for Device specific parameter ranges.

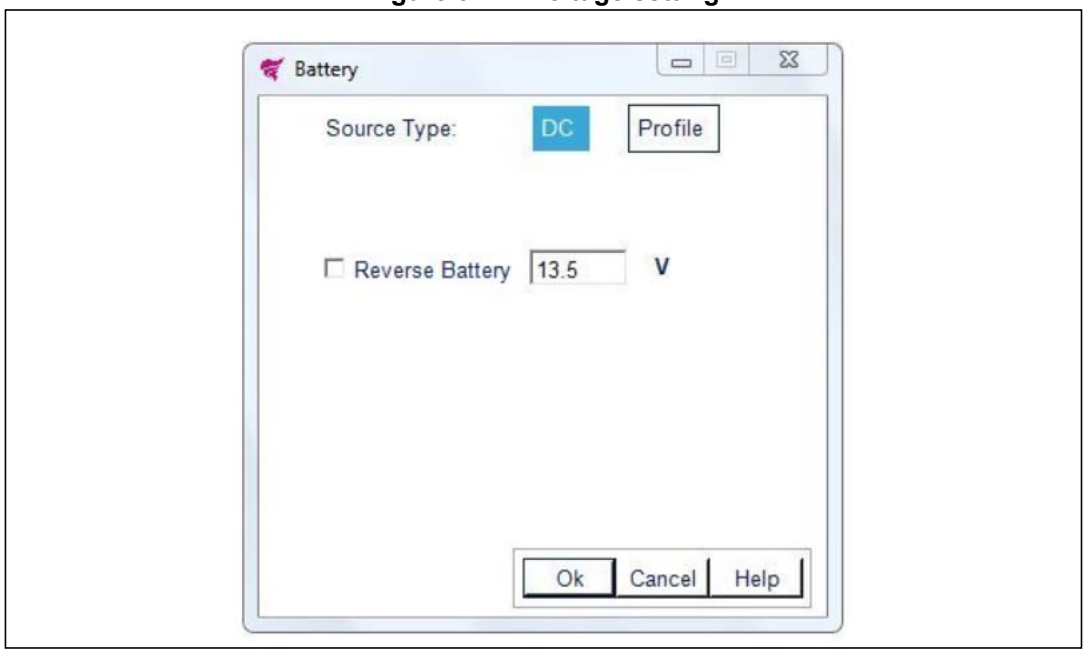
2.2 Nominal voltage selection and setting

Click on the Battery symbol (Nom.Volt.) in the main window, a new frame opens (*Figure 3*) where the nominal operating supply voltage can be modified.

Select "DC" or "Profile" to set the Source voltage.

2.2.1 DC voltage setting

Figure 3. DC voltage setting



Change the value manually by inserting it in the numeric box.

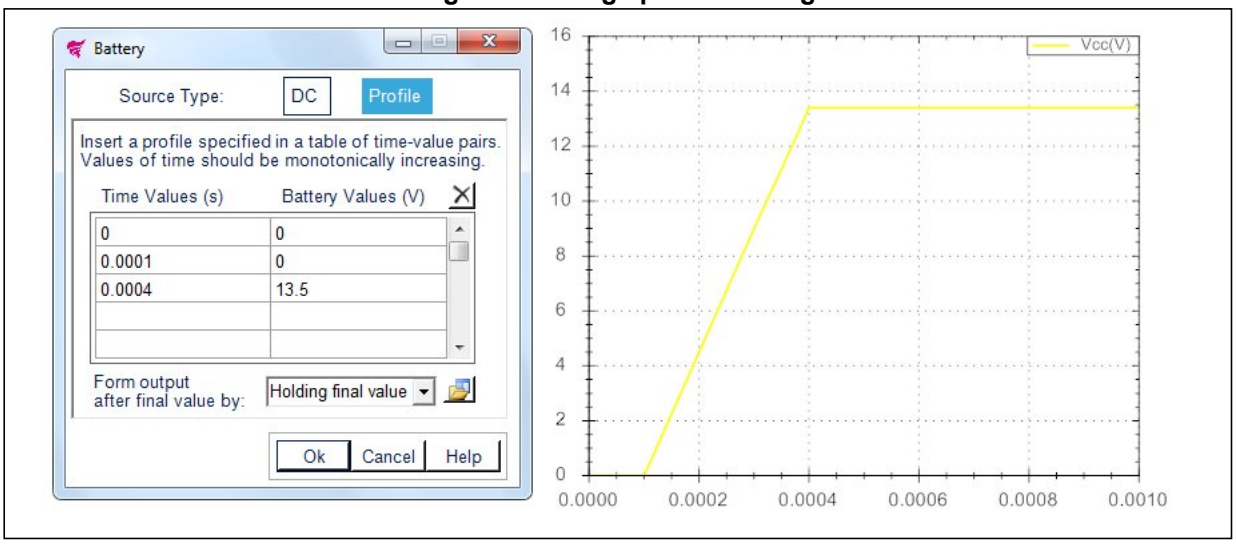
Values outside the boundaries are automatically restored to the max/min allowed value.

Check "Reverse Battery" for negative voltage (HS, LS only).

2.2.2 Voltage profile setting

Click Profile, a new frame opens (*Figure 4*) allowing to insert a custom profile by specifying time-values pairs in the corresponding fields or load values from a .txt/.csv file (time values in the 1st column and Voltage values in the 2nd, max 150 rows).

Figure 4. Voltage profile setting



Define the output after the final value by:

- Cyclic repetition (Voltage values sequence is repeated continuously during the simulation)
- Holding final value (Last voltage value is maintained throughout the simulation).

Time values must be inserted as a increasing monotonic sequence (i.e., Tn+1>Tn).

- Click to clear table
- Click to load values from file

- Click **OK** to accept changes
- Click **Cancel** to restore previous values
- Click **Help** to read the help file on this topic

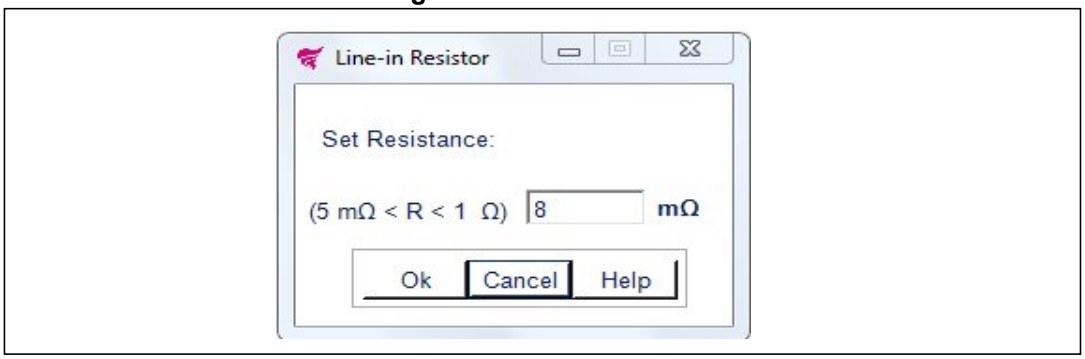
2.3 Line and sense resistances

Click on the Resistors symbols in the main window to change Line-in, Line-out and Sense resistances for analog devices only.

2.3.1 Line-in resistor

Change the value manually by inserting it in the numeric box (*Figure 5*).

Figure 5. Line-in resistor



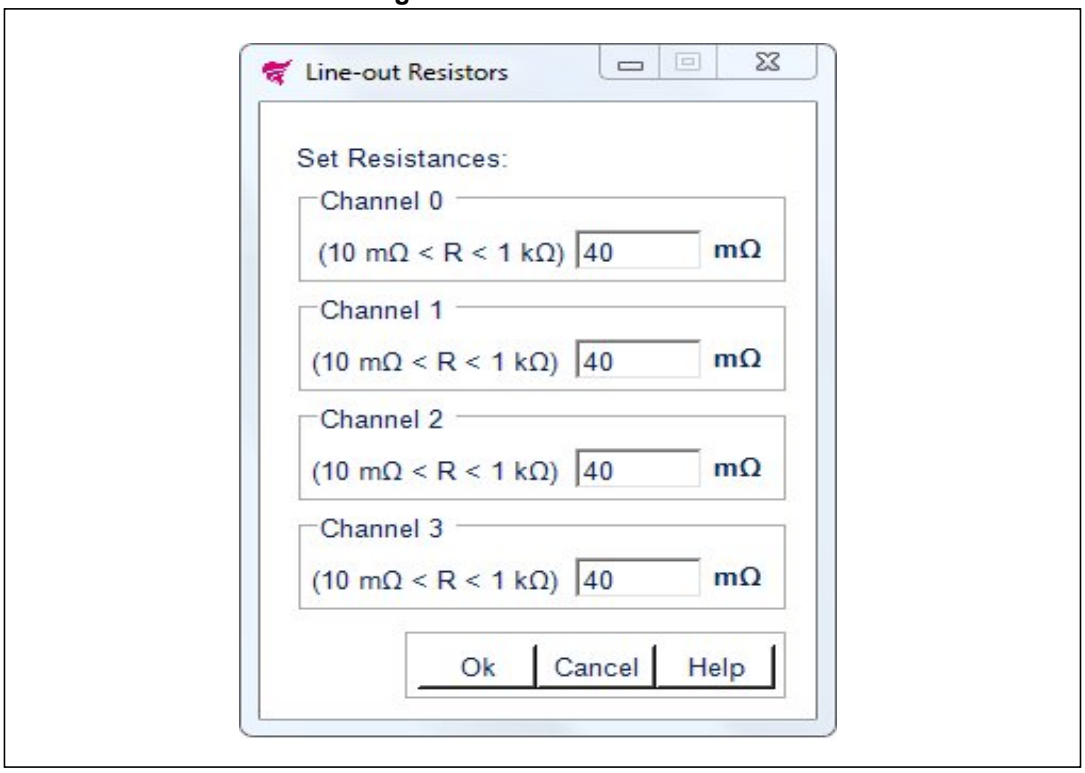
Values outside the boundaries are automatically restored to the maximum allowed value.

Accepted values are between 5 mΩ and 1 Ω.

2.3.2 Line-out resistors

Change the values manually by inserting them in the numeric boxes (*Figure 6*).

Figure 6. Line-out resistors



Values outside the boundaries are automatically restored to the max/min allowed value.

Accepted values are between 10 mΩ and 1 kΩ.

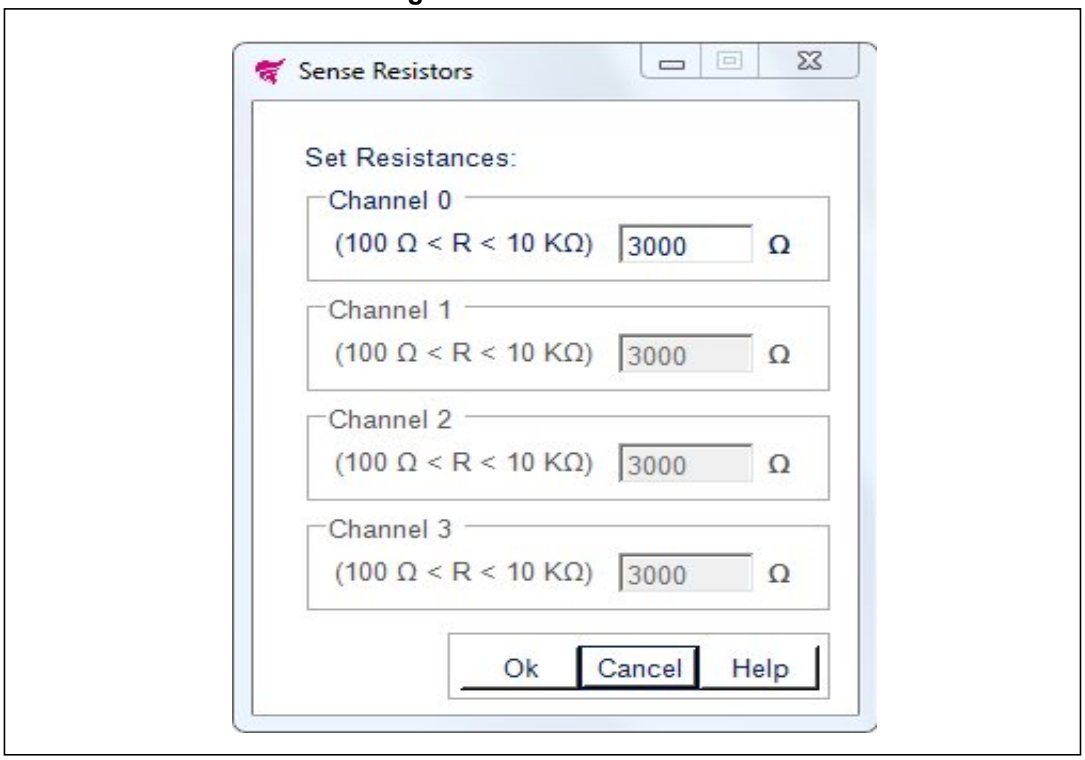
The number of enabled Line-in resistors is equal to the number of output channels for the selected device.

2.3.3 Sense resistors

Only for analog current sense devices, Sense resistors values can be set to simulate current sense diagnostic voltage conversion (*Figure 7*).

Change the values manually by inserting them in the numeric boxes.

Figure 7. Sense resistors



Values outside the boundaries are automatically restored to the max/min allowed value. Accepted values are between 100 Ω and 10 kΩ.
The number of enabled sense resistors is equal to the number of output channels for the selected device.

- Click **OK** to accept changes
- Click **Cancel** to restore previous values
- Click **Help** to read the help file on this topic

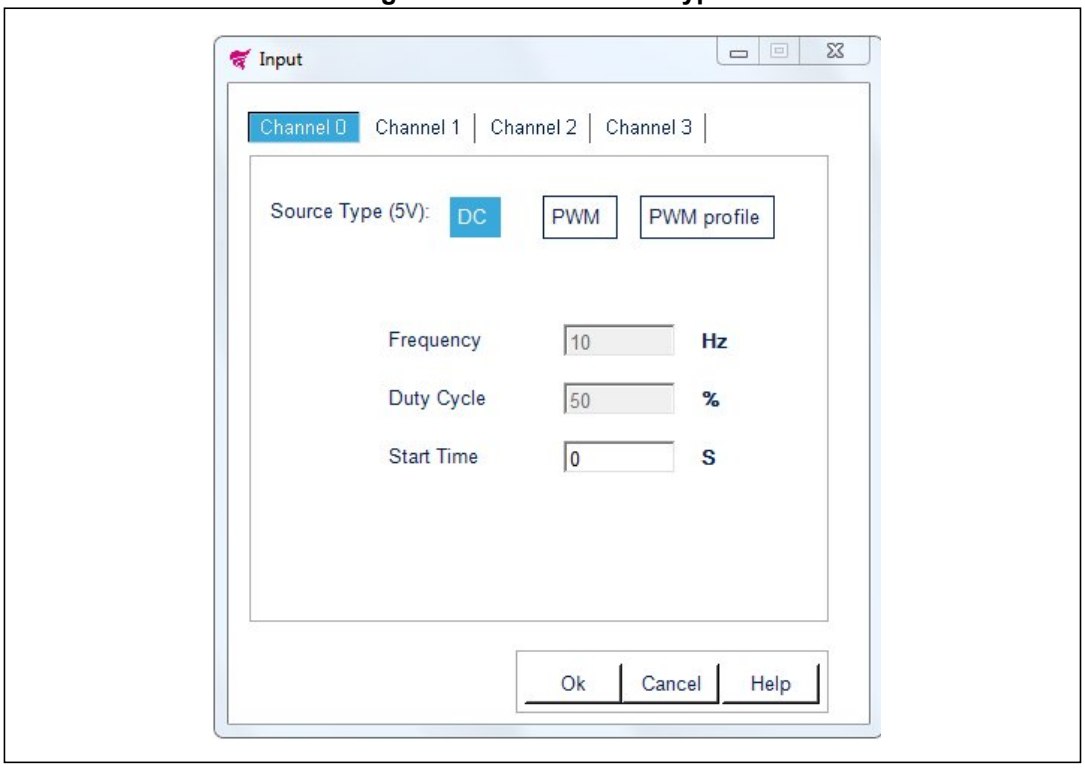
Resistance values are reported in the main window.

2.4 Control source

Click on the Control Voltage symbol (Vin) in the main window, a new frame opens where the control type for each channel can be selected (*Figure 8*).
The number of visible Tabs is equal to the number of output channels of the selected device.

2.4.1 Control source type

Figure 8. Control source type

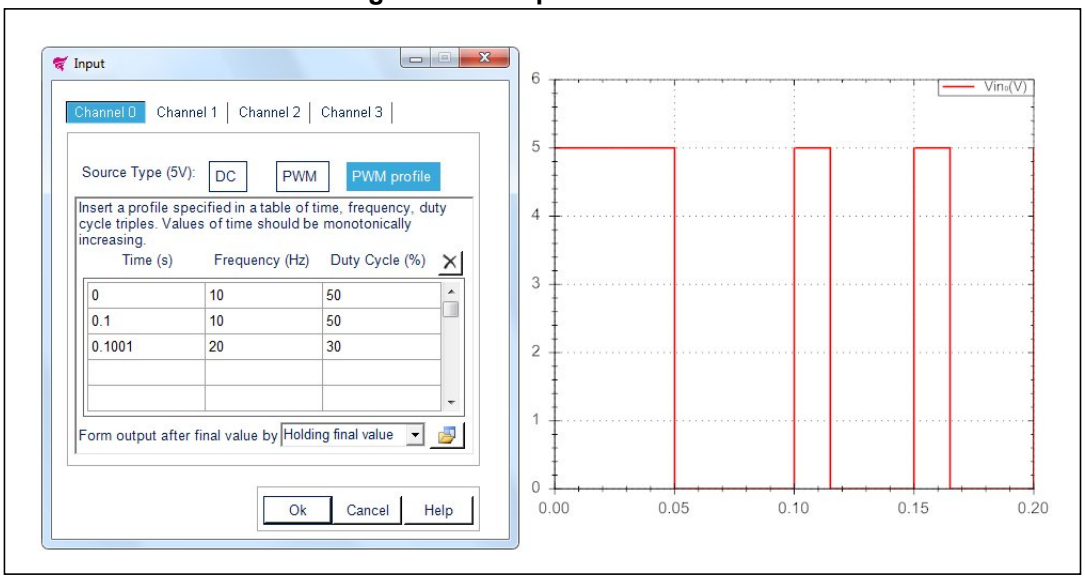


- Selecting "DC" Source type a 5 V DC source voltage is applied to the "In" pin, starting from "Start Time".
- Selecting "PWM" Source type a 5 V square wave source voltage is applied to the "In" pin, with a specific "Frequency" and "Duty Cycle", starting from "Start Time".
- Selecting “PWM” profile Source type a new frame opens (*Figure 9*) in order to insert a custom profile by specifying time, frequency and duty cycle in the corresponding fields or load values from a .txt/.csv file (time values in the 1st column, frequency in the 2nd and duty cycle values in the 3rd, max 150 rows).

Change the values manually by inserting them in the numeric boxes.

Values outside the boundaries are automatically restored to the max/min allowed value.

Figure 9. PWM profile selection



Define the output after the final value by:

- Cyclic repetition (Frequency and duty cycle values sequence is repeated continuously during the simulation)
- Holding final value (Last frequency and duty cycle values are maintained throughout the simulation)

Time values must be inserted as an increasing monotonic sequence (i.e., Tn+1>Tn).

Click ✕ to clear table

Click 📂 to load values from file

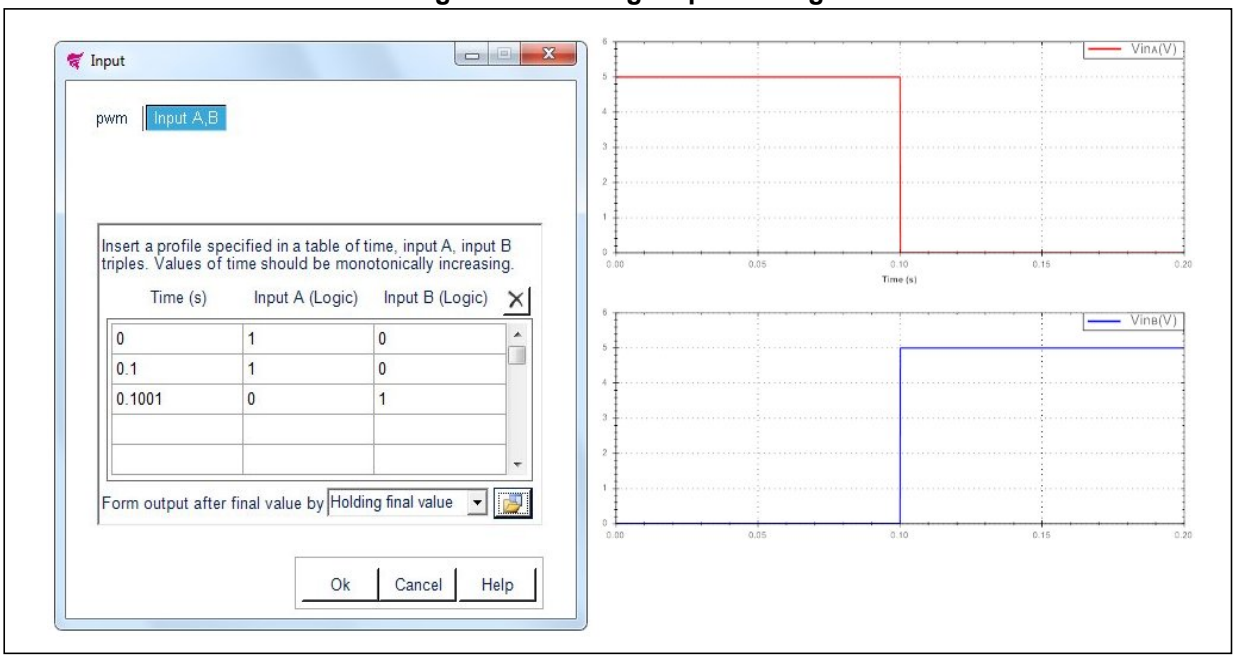
PWM and PWM Profile for the selected channel are not available if Current or Power Profile is selected.
PWM and PWM Profile disable the Current and Power Profile selection for the selected channel.

2.4.2 H-Bridge input setting

Click on the "PWM, Vin A,B" in the main window then select “Input A,B” and a new frame (*Figure 10*) allows inserting a custom profile by specifying time, Input A and Input B in the corresponding fields or load values from a .txt/.csv file (time values in the 1st column, Input A in the 2nd and Input B values in the 3rd, max 150 rows).
Define the output after the final value by selecting:

- Cyclic repetition (Input A and Input B values sequence is repeated during the simulation)
- Holding final value (Last Input A and Input B values are maintained throughout the simulation)

Figure 10. H-Bridge input setting



Time values must be inserted as a increasing monotonic sequence (i.e., Tn+1>Tn). Input A and Input B are logical values:

- Set Input A = 1 and Input B = 0 to activate HS_1-LS_2 diagonal
- Set Input A = 0 and Input B = 1 to activate HS_2-LS_1 diagonal

Click ✕ to reset table

Click to load values from file

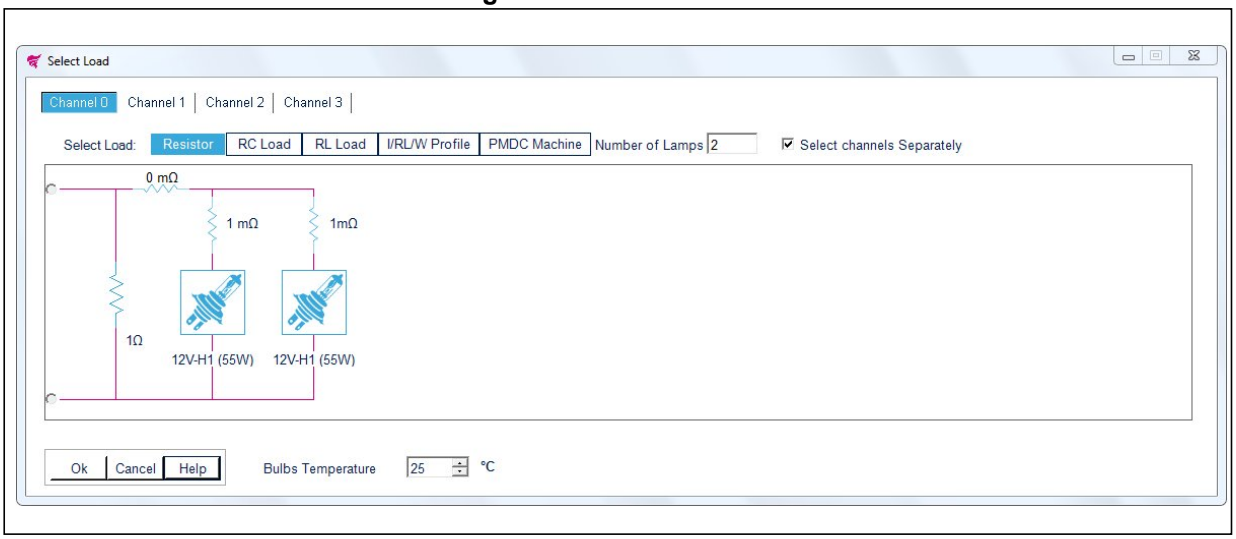
Changes in Input selection disable Current Profile selection
Reset table to enable Current Profile selection with DC source type.

- Click **OK** to accept changes
- Click **Cancel** to restore previous values
- Click **Help** to read the help file on this topic

2.5 Load selection

Click on the Load icon in the main window, a new frame opens (*Figure 11*) where load parameters for each output channel can be selected.
Number of visible Tabs is equal to the number of output channels for the selected device.

Figure 11. Load selection



Select one of the following loads:

- Resistor (*Figure 12*)
- RC (*Figure 13*)
- RL (*Figure 14*)
- Current, RL or Power Profile (*Figure 15*)
- PMDC Machine (*Figure 18*)

From the counter "Number of Lamps" a maximum of 10 lamps per channel can be selected. If "Select channels Separately" is unchecked, the same load is selected for each channel.

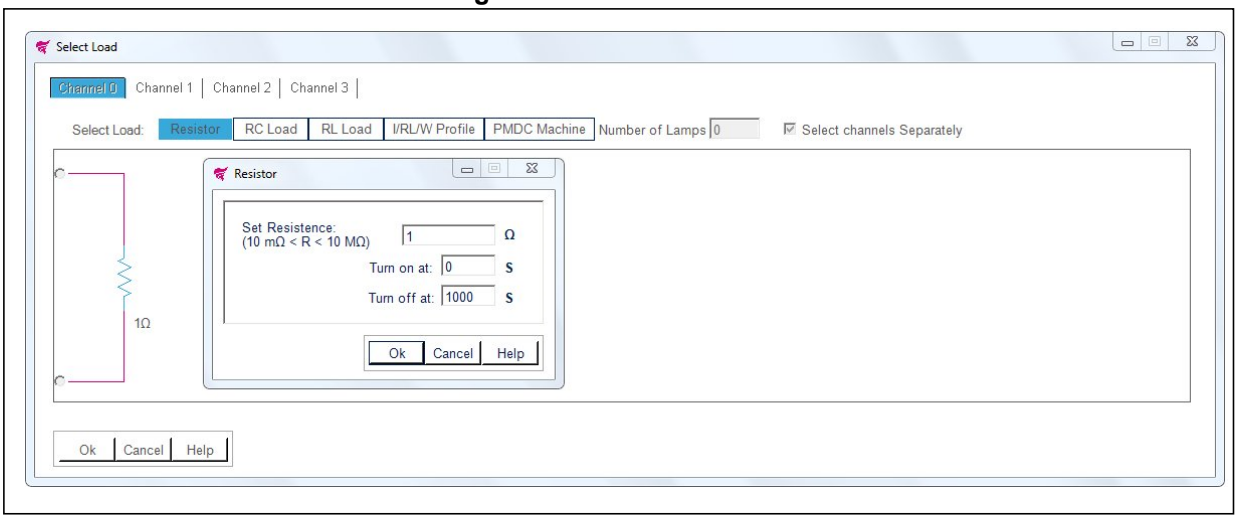
The values of each component in the load can be manipulated. Click on the components icons to change the values.
If no load is selected, an open circuit symbol appears.

- Click **OK** to accept changes
- Click **Cancel** to restore previous layout
- Click **Help** to read the help file on this topic

2.5.1 Resistor as load

Click "Resistor" button to select a Resistive load and then click on the Resistor symbol in the Layout to change its value.

Figure 12. Resistor as load



Change the values manually by inserting them in the numeric boxes.
Values outside the boundaries are automatically restored to the max/min value allowed.
Accepted values are between 10mΩ and 10MΩ for resistance.

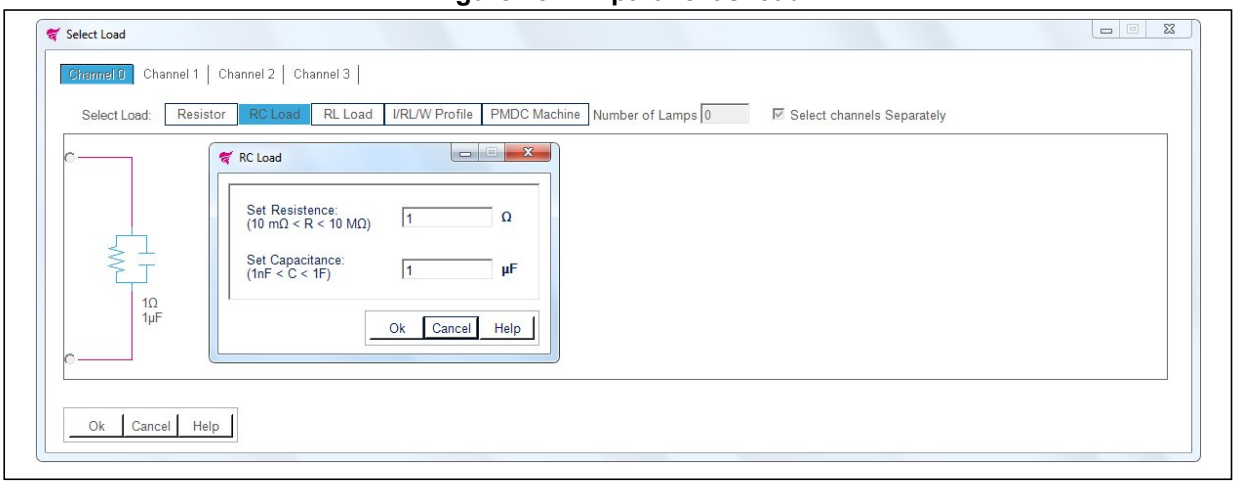
- "Turn On" time: connect Resistor during simulation at a specific time
- "Turn Off" time: disconnect Resistor during simulation at a specific time

- Click **OK** to accept changes
- Click **Cancel** to restore previous values
- Click **Help** to read the help file on this topic

2.5.2 RC parallel as load

Click "RC Load" button to select an RC parallel load and then click on the Capacitor symbol in the layout to change its value

Figure 13. RC parallel as load



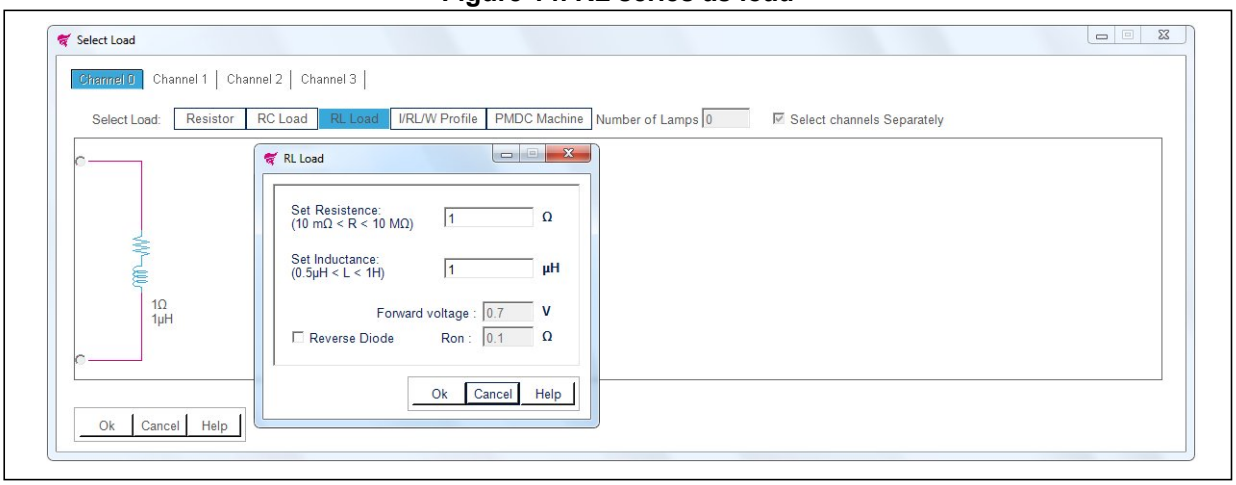
Change the values manually by inserting them in the numeric boxes.
Values outside the boundaries are automatically restored to the max/min allowed value.
Accepted values are between 10 mΩ and 10 MΩ for resistance and between 1 nF and 1 F for Capacitance.

- Click **OK** to accept changes
- Click **Cancel** to restore previous values
- Click **Help** to read the help file on this topic

2.5.3 RL series as load

Click "RL Load" button to select an RL series load and then click on the Inductor symbol in the layout to change its value

Figure 14. RL series as load



Change the values manually by inserting them in the numeric boxes.
Values outside the boundaries are automatically restored to the max/min value allowed.
Accepted values are between 10 mΩ and 10 MΩ for resistance and between 0.5 µH and 1 H for Inductance.

Check "Reverse Diode" to select a reverse diode in parallel with RL load.
Accepted values are between 10 mV and 1 V for "Forward voltage" and between 1 mΩ and 1 Ω for "Ron".
Reverse Diode is not available for H-Bridge Devices.

- Click **OK** to accept changes
- Click **Cancel** to restore previous values
- Click **Help** to read the help file on this topic

RL Load disables Lamp selection for selected channel.

2.5.4 Current, RL and device power loss profile as load

Channel 1:

- Click "I/RL/W Profile" button to select an output current (Iout), an RL Load or a Device Power Loss (Vds • Iout) Profile and then click on the corresponding I/RL/W symbol in the layout to change its value.

Channels 2...4:

- Click "I/RL Profile" button to select a an output current (Iout) or an RL Load Profile and then click on the corresponding I/RL symbol in the layout to change its value.

Select Output current (*Figure 15*), RL Load (*Figure 16*) or Device Power Loss Profile (1st channel) (*Figure 17*) from the pop-up menu.

Figure 15. Output current

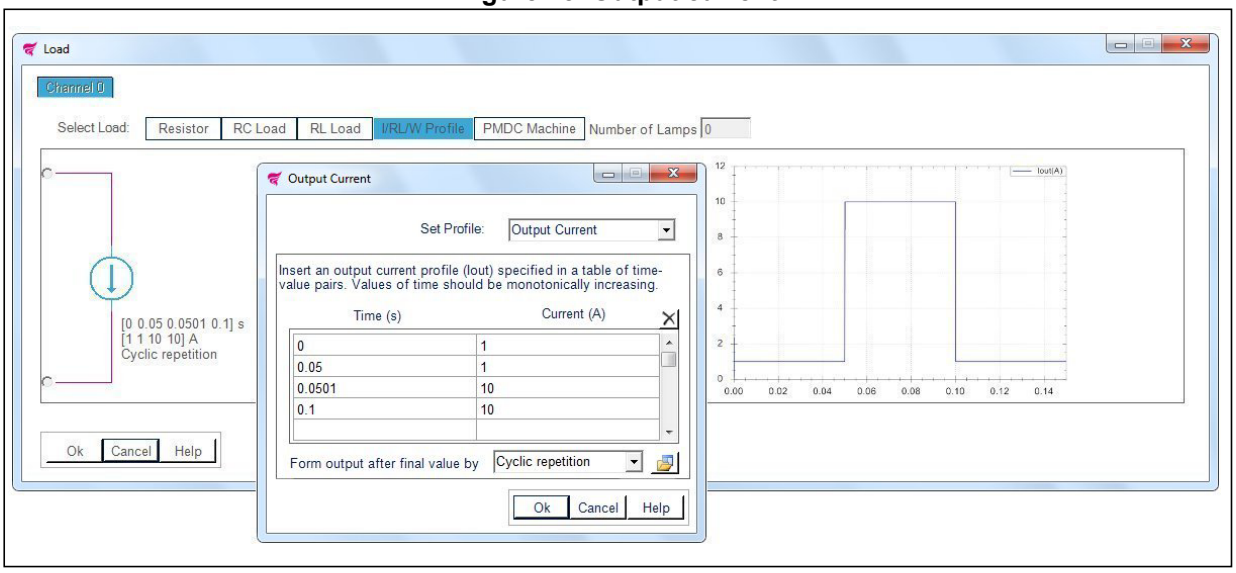


Figure 16. RL load

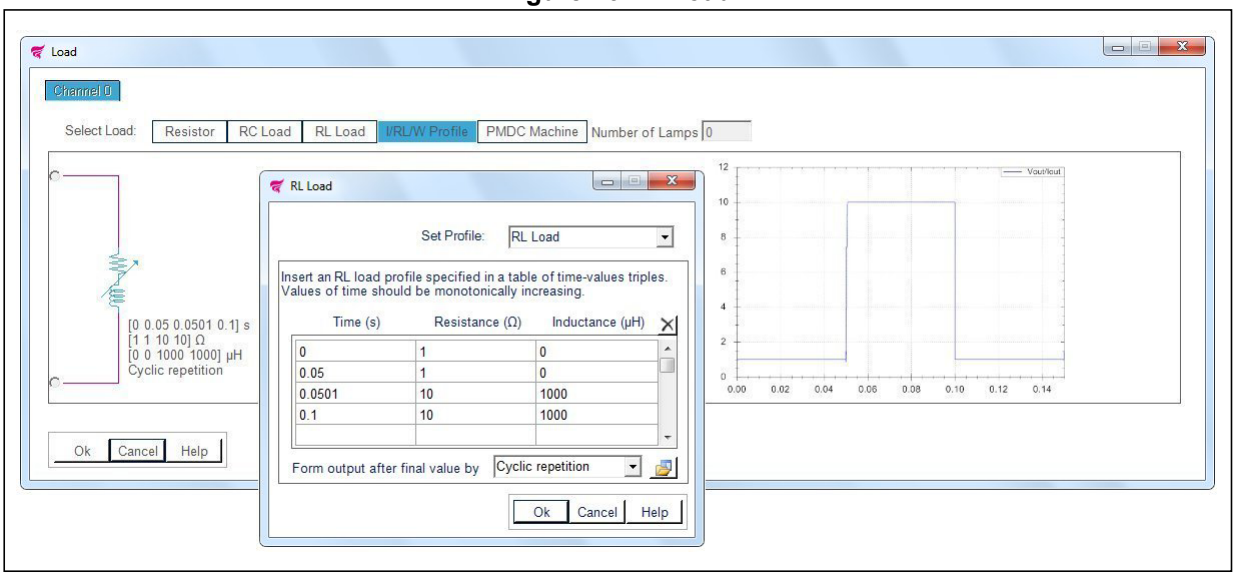
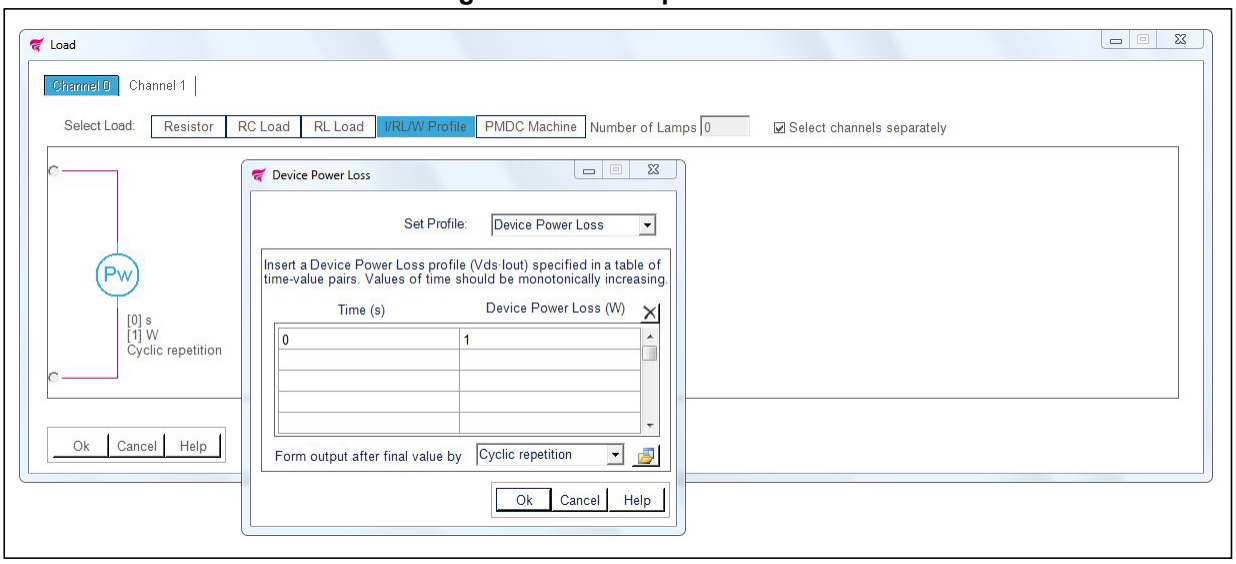


Figure 17. Device power loss



Insert a custom profile by specifying time-value pairs or triples in the corresponding fields or load values from a .txt/.csv file (time values in the 1st column; Current, Resistance or Power values in the 2nd; Inductance values in the 3rd (Only for RL Load profile), max 150 rows).

For Resistance Profile insert 0 in the Inductance fields.
Define the output after the final value by:

- Cyclic repetition (Current, RL Load or Power values sequence is repeated continuously during the simulation)
- Holding final value (Last Current, RL Load or Power value is maintained throughout the simulation).

Time values must be inserted as an increasing monotonic sequence (i.e., Tn+1>Tn).
Click ✕ to clear table.

Click to load values from file

Establishing a Current Profile, disables PWM selection for the selected channel, Input AB for H-Bridge devices (use negative currents to invert H-Bridge polarity).
Establishing a Power Profile (1st channel) disables PWM, channels and H-Bridge selection.

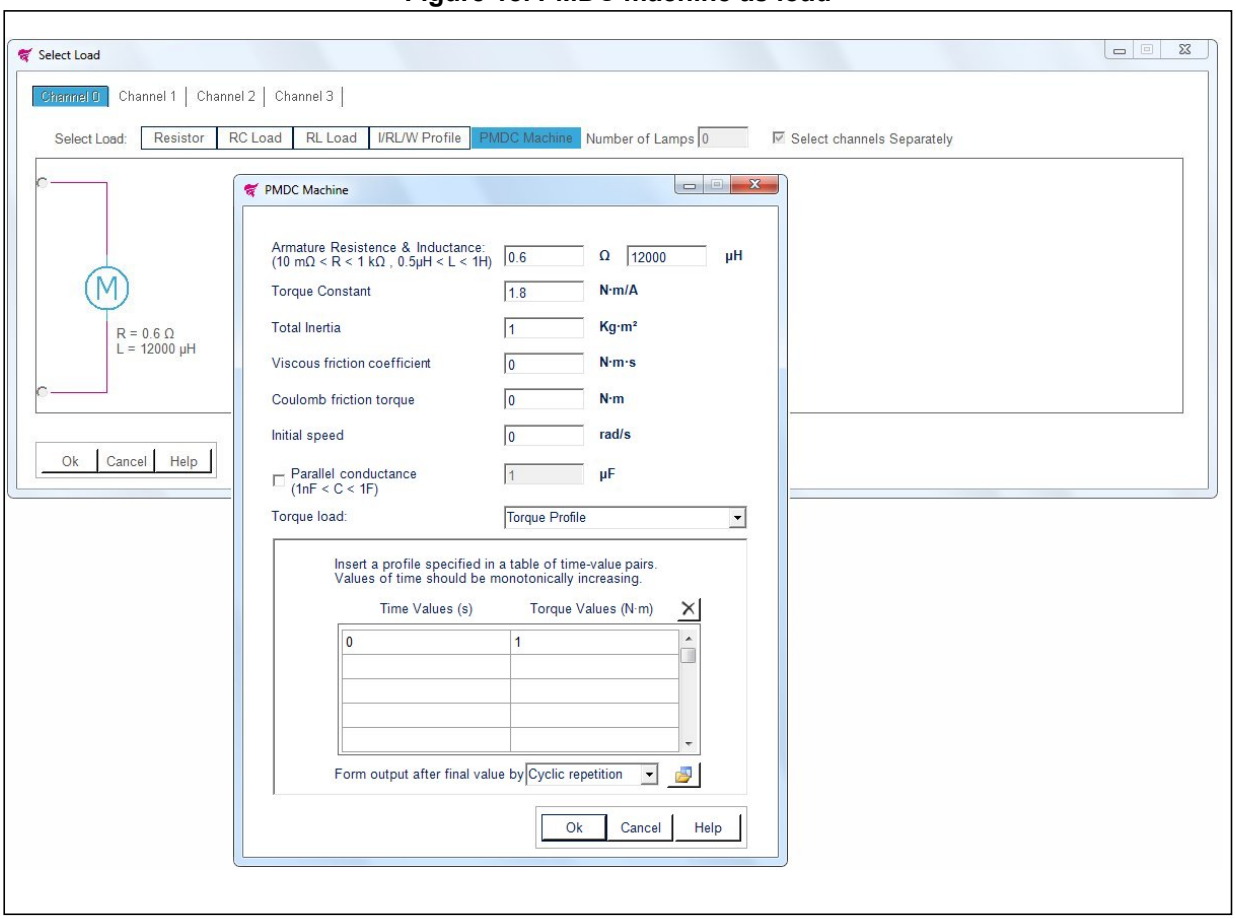
- Click **OK** to accept changes
- Click **Cancel** to restore previous values
- Click **Help** to read the help file on this topic

A current (*Figure 15*), RL Load (*Figure 16*) or Power (*Figure 17*) symbol appears in the load layout.
I/RL/W Profile disables Lamp selection for selected channel.

2.5.5 PMDC machine as load

Click "PMDC Machine" button to select a Permanent Magnet DC Machine and then click on the PMDC symbol in the Layout to change its parameters.
Change the values manually by inserting them in the numeric boxes.
Values outside the boundaries are automatically restored to the max/min value allowed.

Figure 18. PMDC machine as load



PMDC Machine parameters:

- Armature Resistance (Ω): The resistance of the winding
- Armature Inductance (H): The inductance of the winding
- Torque constant (N·m/A): The torque per current constant
- Total inertia (kg·m^2): The total inertia momentum of the motor and the load
- Viscous friction coefficient (N·m·s): The total friction coefficient
- Coulomb friction torque (N·m): The total friction torque constant
- Initial speed (rad/s): The initial rotation speed

Check parallel conductance to insert a capacitance in parallel to PMDC machine.

PMDC Specifical signals:

- T_L (N·m): The electromechanical torque load at the input of the DC machine
- T_e (N·m): The electromechanical torque developed by the DC machine
- ω (rad/s): The rotational speed of the DC machine

Mechanical input (Torque Load):

- Hold Motor: Hold motor position (T_L=T_e and ω=0)
- Open Load: No electromechanical torque load at the input (T_L=0)
- Torque Profile: Insert a custom profile by specifying time-values pairs in the corresponding fields or load values from a .txt/.csv file (time values in the 1st column and torque values in the 2nd, max 150 rows).

Define the output after the final value by:

- Cyclic repetition (Torque values sequence is repeated continuously during the simulation)
- Holding final value (Last Torque value is maintained throughout the simulation).

Time values must be inserted as a increasing monotonic sequence (i.e., Tn+1>Tn).

Click ✕ to reset table

Click 📂 to load values from file

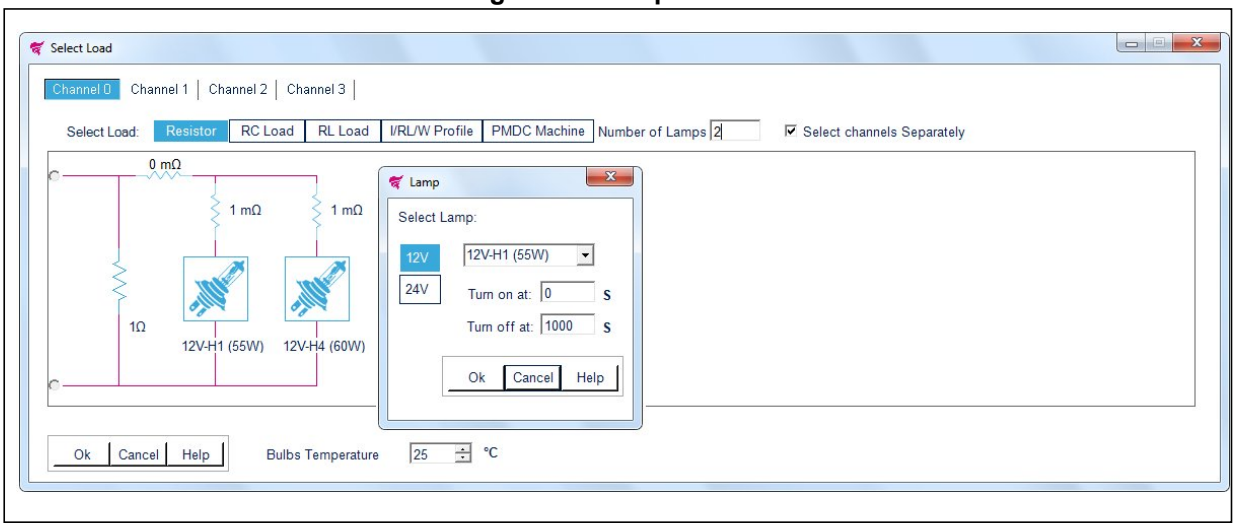
- Click **OK** to accept changes
- Click **Cancel** to restore previous values
- Click **Help** to read the help file on this topic

PMDC Machine disables Lamp selection for selected channel.

2.5.6 Lamp as load

Select up to 10 lamps from the "Number of Lamps" counter and then click on a Lamp icon in the layout to change the values.

Figure 19. Lamp as load



Lamps can be placed alone or in parallel to a resistor or a capacitor and they can have different values.
Select a lamp from the drop down menu, according to its voltage and power requirements.

Lamp parameters:

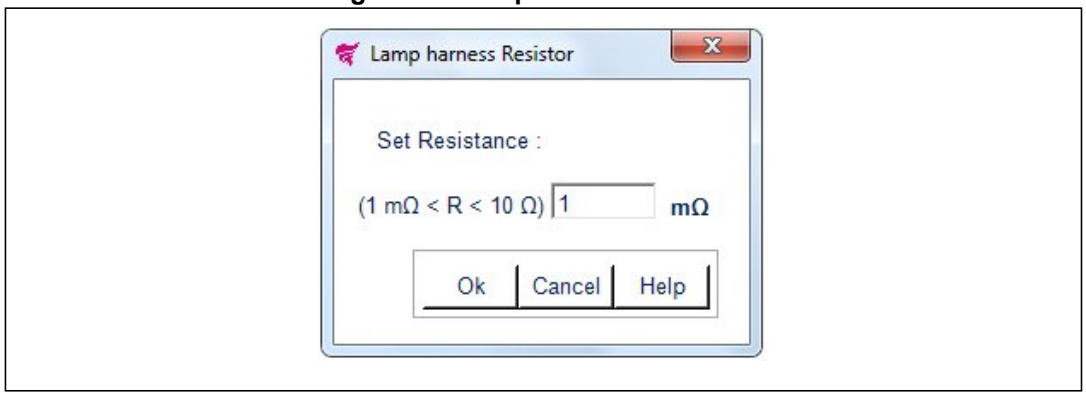
- "12 V": select 12 V lamps
- "24 V": select 24 V lamps
- "Turn On" time: connect lamp during simulation at a specific time
- "Turn Off" time: disconnect lamp during simulation at a specific time
- "Bulbs Temperature": Initial filament temperature (°C). Accepted values are between -40 °C and 1000 °C. This value is common to all lamps inserted

- Click **OK** to accept changes
- Click **Cancel** to restore previous values
- Click **Help** to read the help file on this topic

Lamp harness resistors

Click on the Resistors symbols in the layout to set a wiring harness lamp resistance for each lamp

Figure 20. Lamp harness resistors



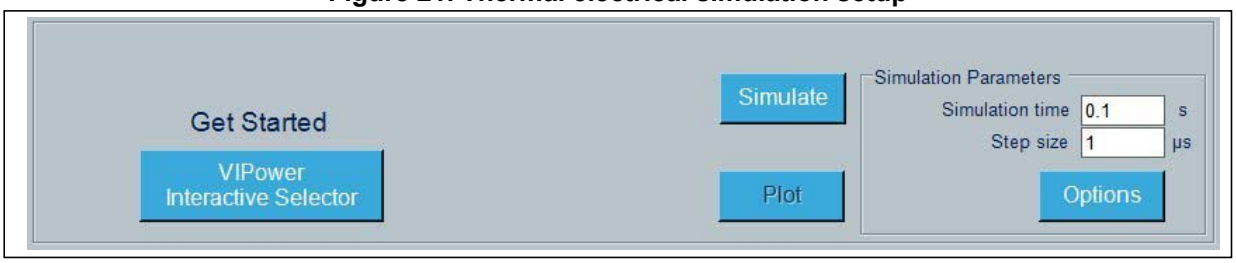
Change the value manually by inserting it in the numeric box.
Values outside the boundaries are automatically restored to the max/min allowed value.
Accepted values are between 1 mΩ and 10 Ω.

- Click **OK** to accept changes
- Click **Cancel** to restore previous values
- Click **Help** to read the help file on this topic

2.6 Thermal electrical simulation setup

- Choose simulation time and then Click "Simulate" button to Start.

Figure 21. Thermal electrical simulation setup



"Step Size" is automatically selected based on the Device/Load configuration.

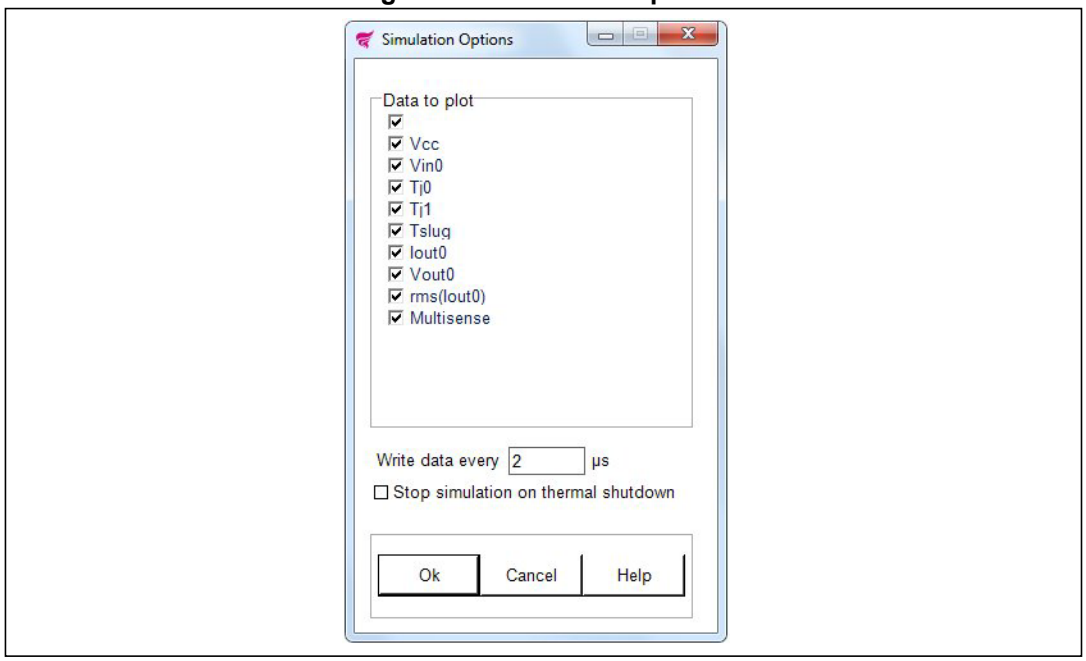
Changing Step Size affects the simulation speed and accuracy:

For best precision set a lower Step Size. For faster simulation set a higher Step Size.

A higher Step Size may cause convergence problems.

Click options for more simulation parameters.

Figure 22. Simulation options



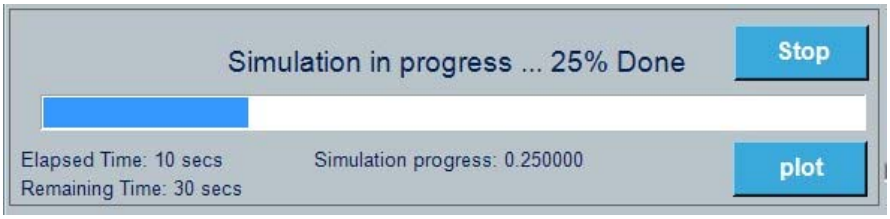
- Select data to plot: checked items will be available for graphical simulation results (default all).
- Select "Write data every" as a multiple of "Step Size".
- Check the box "Stop simulation on thermal shutdown" to interrupt the simulation when Tj is higher than thermal shutdown temperature.

"Write data every" is automatically selected based on "Simulation Time" and "Step Size", in order to optimize resolution and memory requirements.

Lower values do not affect simulation accuracy but may cause memory problems.

A progress bar indicates the state of the simulation.

Figure 23. Simulation progress bar



Computation time is principally a function of the kind of load (inductive loads take a longer time), "Simulation Time" and "Step Size" parameters.

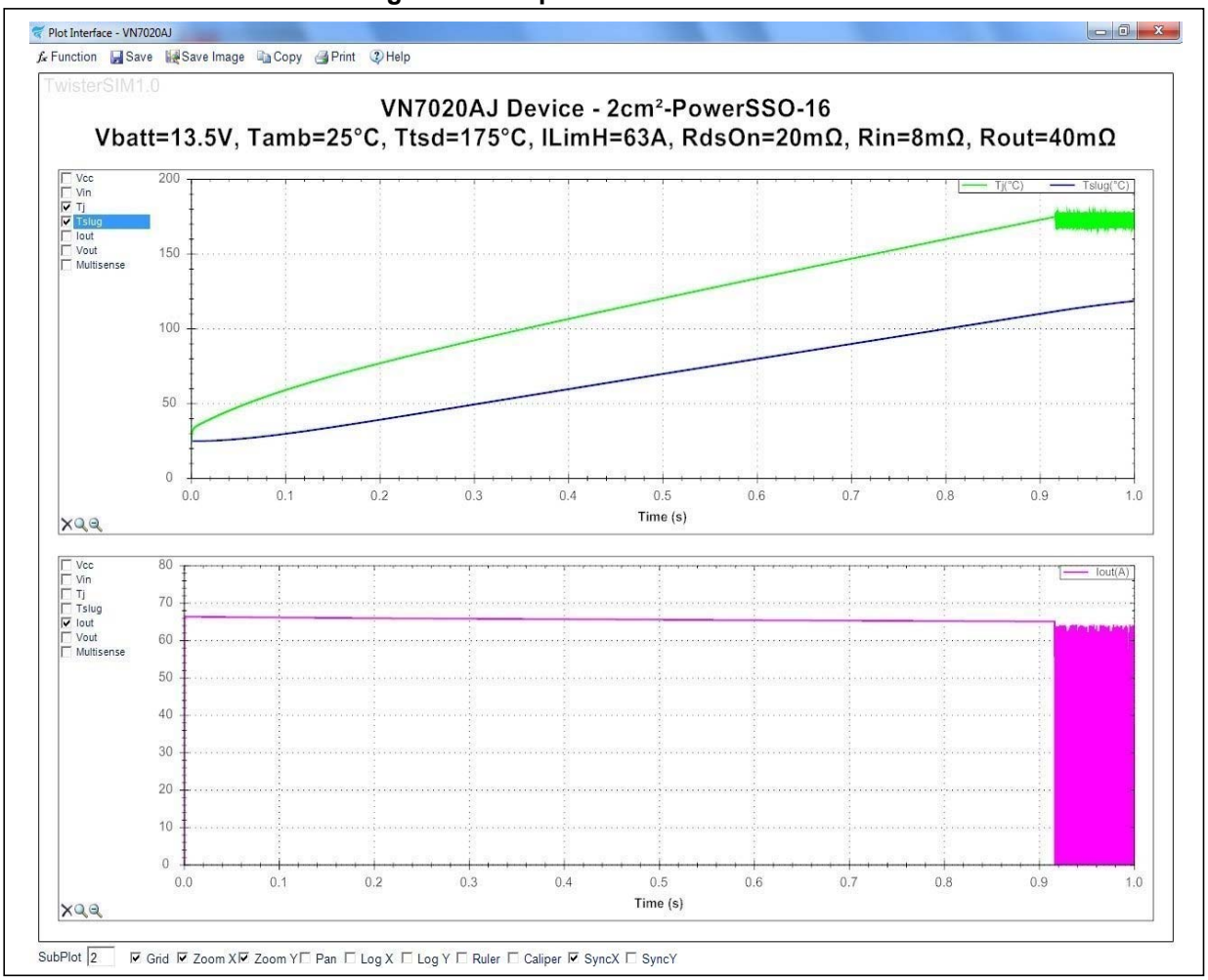
At the end of simulation a plot window shows simulation results.

- Click "Stop" button to stop simulation
- Click "Plot" button to show or reopen Plot window
- Click "Plot" button during simulation to show partial results: A notice in the plot informs about the state of simulation.

2.7 Graphical simulation results

When the simulation is over, a new window is opened to show the simulation results.

Figure 24. Graphical simulation results

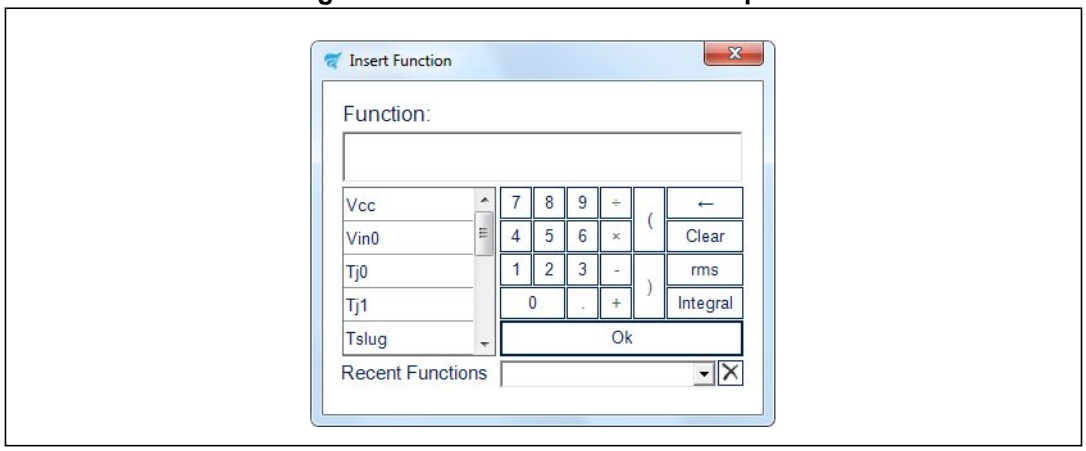


- Left click on the signals list checkbox to add or remove them in the plot (*Figure 24*)
- Right click on the signals to change color, line style or signal name
- Right click on the plot to change titles, un-zoom, un-pan, copy save or print the image
- Change the Subplots number to show simulation results in up to 4 subplots.

- Check "Grid" to show grid lines
- Check "Zoom X" to enable Horizontal zoom
- Check "Zoom Y" to enable Vertical zoom
- Check "Pan" to interactively move the plot view, left clicking on the plot
- Check "Log X" to set logarithmic X axes scale
- Check "Log Y" to set logarithmic Y axes scale
- Check "Ruler" to show a ruler in the plot.
- Check "Caliper" to show a caliper in the plot.
- Check "SyncX" to synchronize Subplots X axes
- Check "SyncY" to synchronize Subplots Y axes

- Click to save the results to a data file for exporting to spreadsheets or a generic text file
- Click to save a picture of the plot
- Click to copy a picture of the plot
- Click to print a picture of the plot
- Click to show this help
- Click to restore the scale ranges to default values
- Click to restore the scale ranges to the values before last zoom
- Click to add signals combination or function to plot:

Figure 25. Create a new function to plot



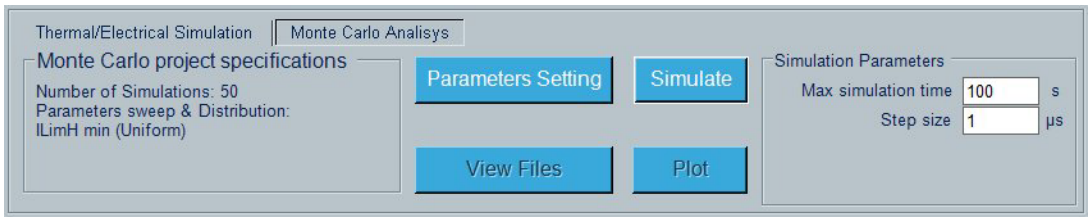
Insert a signal combination or function, clicking or typing in the function field.

- Click "←" for backspace
- Click "Clear" to clear text.
- Click "rms" for rms calculation.
- Click "Integral" for integral calculation.
- Click on the listbox to select a recent function
- Click "OK" to insert function.

2.8 Monte Carlo analysis

The Monte Carlo analysis performs multiple runs of the selected layout, statistically changing selected parameters in the defined range.

Figure 26. Monte Carlo analysis simulation setup



- Click "Parameters Setting" to set Monte Carlo options.
- Choose simulation time and then Click "simulate" button to Start.

"Step Size" is automatically selected based on the Device/Load configuration.

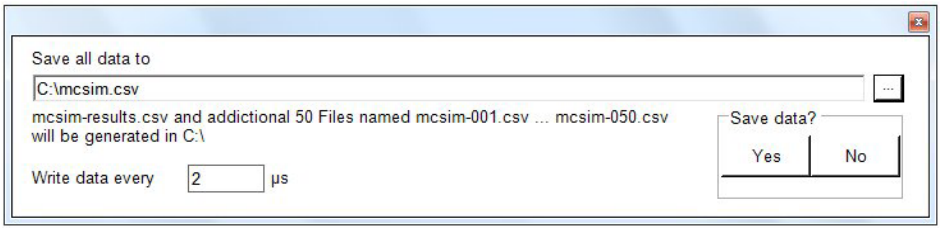
Changing Step Size affects the simulation speed and accuracy:

For best precision set a lower Step Size. For faster simulation set a higher Step Size.

A higher Step Size may cause convergence problems.

After clicking "Simulate" a pop-up appears:

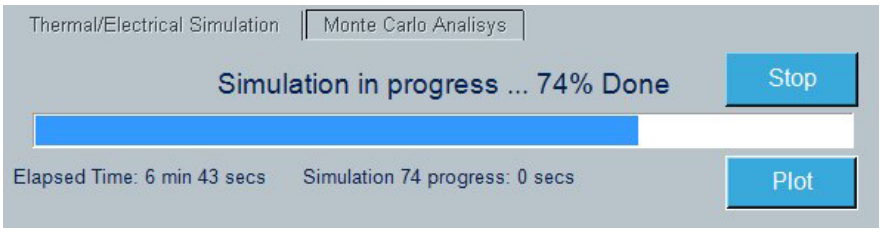
Figure 27. Save Monte Carlo analysis simulation data



- Click ... to change file name
- Click "Yes" to save data for all simulations
  - N+1 files will be generated in the selected folder
  - filename-001 ... filename-N contains complete simulation data, filename-results contains information's for all simulations
  - "Write data every" is automatically selected based on "Simulation Time" and "Step Size", in order to optimize resolution and memory requirements.
  - Lower values do not affect simulation accuracy but may cause memory problems.
- Click "No" to simulate without saving all data.

When simulation starts a progress bar indicates the state of the simulation.

Figure 28. Monte Carlo analysis wait for simulation



Computation time is principally a function of the number of simulations, the kind of load (inductive loads take a longer time), "Simulation Time" and "Step Size" parameters.

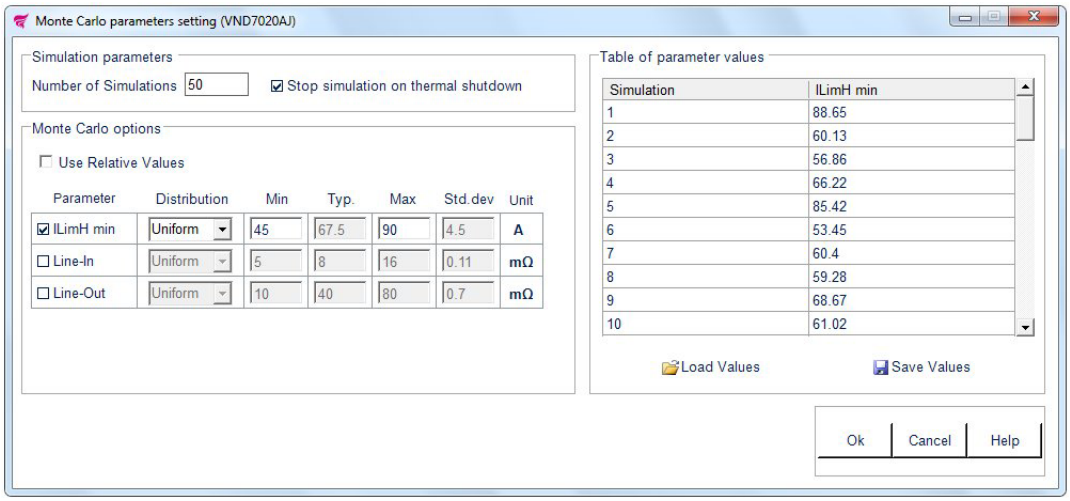
At the end of simulation a plot window shows simulation results.

- Click "Stop" button to stop simulation.
- Click "Plot" button to show or reopen Plot window.
- Click "Plot" button during simulation to show partial results: a notice in the plot informs about the state of simulation.
- Click "View Files" at the end of simulation to open folder containing simulation data (if any).

2.8.1 Monte Carlo parameters setting

Click "Parameters Setting" in Monte Carlo Analysis tab to set Monte Carlo options.

Figure 29. Monte Carlo parameters setting



Set Number of Simulations by inserting it in the numeric box.

Values outside the boundaries are automatically restored to the max/min allowed value.

Accepted values are between 1 and 1000.

Check the box "Stop simulation on thermal shutdown" to interrupt the simulation when Tj is higher than thermal shutdown temperature.

Check parameter name to perform Monte Carlo analysis on selected item

Select one of the following distributions:

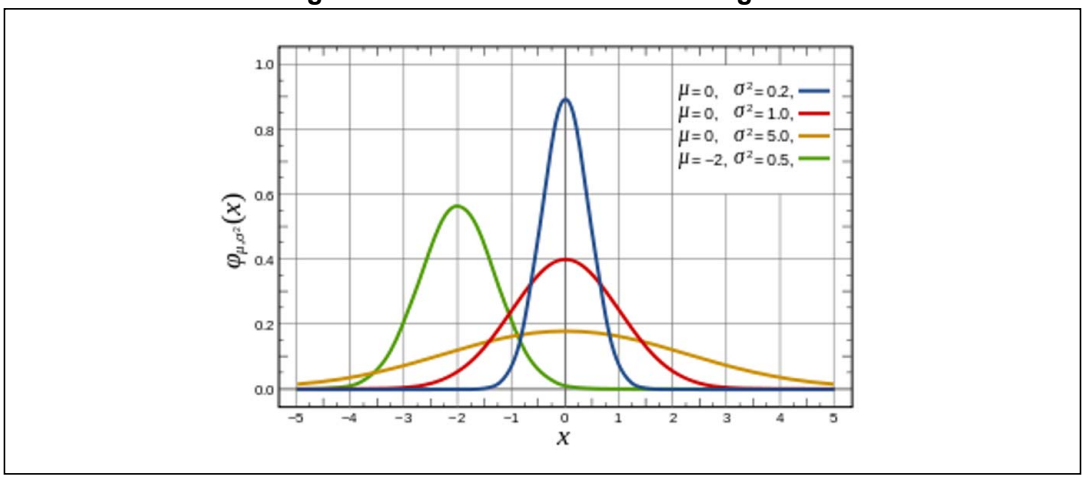
- Uniform (Generates uniformly distributed values in the range Min-Max)
- Gaussian (Generates Gaussian distributed values in the range Min-Max)

The probability density of the Gaussian distribution is:

$$f(x \mid \mu, \sigma) = \frac{1}{\sigma\sqrt{2\pi}} \cdot e^{-\frac{(x-\mu)^2}{2\sigma^2}}$$

Were μ is the mean (typical value), σ is the standard deviation and σ^2 is the variance of the distribution.

Figure 30. Gaussian distribution diagram



According to Min, Typ, Max and Std.dev values only a subset of the Gaussian distribution in the range min-max may be generated.

The default values of μ = (max+min)/2 and σ = (max-min)/10 ensures that 99.99% of the population is in the range min-max

A value of σ near to zero approaches typical value, a value of σ = (max-min) approaches a uniform distribution.

Min, Max values can be changed in the parameter range.
Typical value and standard deviation can be changed only for Gaussian distribution.
Refer to the relevant datasheet for device specific parameter ranges.

Check "Use Relative Values" to show relative values.

Selected parameters values will vary according to the formula

Parameter_Min + Relative_Value * (Parameter_Max - Parameter_Min),
0 ≤ Relative_Value ≤ 1

Click to load table values from text or csv file

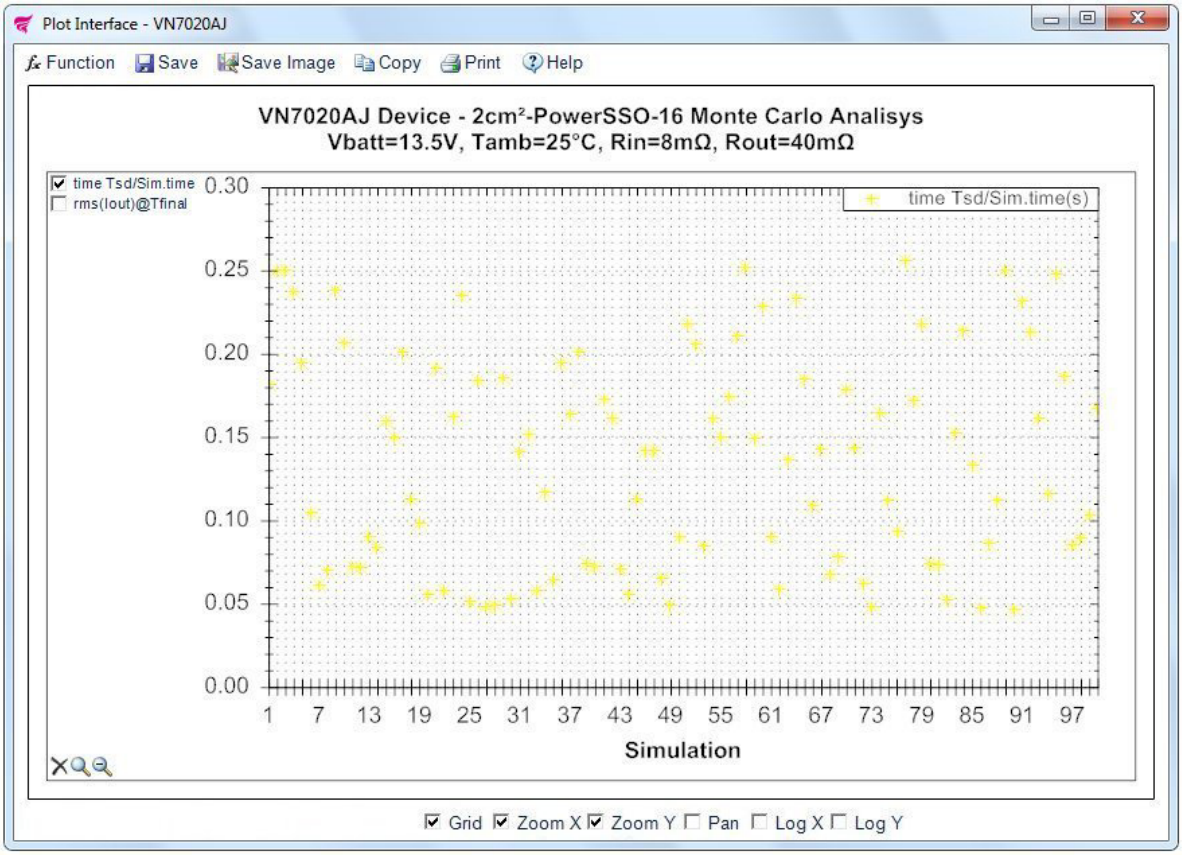
Click to save table values to text or csv file

- Click **OK** to accept changes.
- Click **Cancel** to restore previous values.
- Click **Help** to read the help file on this topic.

2.8.2 Graphical Simulation Results (MC analysis)

- When the simulation is over, a new window is opened to show the simulation results.

Figure 31. Graphical simulation results (MC analysis)

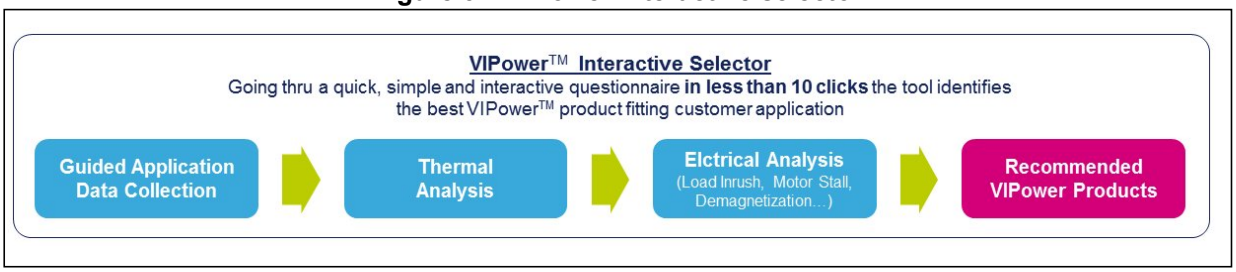


- Left click on the signals list checkbox to add or remove them in the plot.
- Right click on the signals to change color, line style or signal name.
- Right click on the plot to change titles, color, line style, signal name, un-zoom, un-pan, copy save or print the image.
- Check "Grid" to show grid lines.
- Check "Zoom X" to enable Horizontal zoom.
- Check "Zoom Y" to enable Vertical zoom.
- Check "Pan" to interactively move the plot view, left clicking on the plot.
- Check "Log X" to set logarithmic X axes scale.
- Check "Log Y" to set logarithmic Y axes scale.

- Click  to save the results to a data file for exporting to spreadsheets or a generic text file.
- Click  to save a picture of the plot.
- Click  to copy a picture of the plot.
- Click  to print a picture of the plot.
- Click  to show this help.
- Click  to restore the scale ranges to default values.
- Click  to restore the scale ranges to the values before last zoom.
- Click "Plot" button in the main window to show or reopen Plot window.
- Click "Plot" button during simulation to show partial results: a notice in the plot informs about the state of simulation.

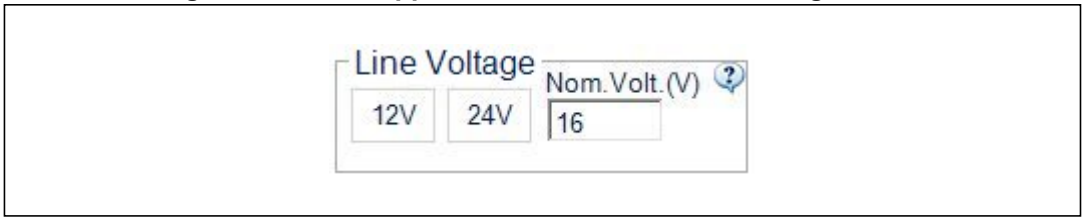
3 VIPower interactive selector overview

Figure 32. VIPower interactive selector



3.1 Select application line and nominal voltage levels

Figure 33. Select application line and nominal voltage levels



Line voltage could be selected between two possible options:

- 12 V: for typical Automotive systems or every application based on 12 V battery line
- 24 V: for Truck or every application based on 24 V battery line

Nominal operating supply voltage is used by simulation engine to perform thermal analysis.

3.2 Select device topology

Figure 34. Select device topology

- In High Side configuration the driver is connected between voltage line and load; the load has the other terminal connected to ground.
- In Low Side configuration the driver is connected between load and ground.
- For unidirectional motors driving either High Side or Low Side configuration can be used
- To drive the motor in both directions, H-Bridge configuration is recommended.

3.3 Select number of channels

Figure 35. Select number of channels

Select how many loads you wish to drive with a single device:

- High Side Switches are available in Single, Dual and Quad channel configurations.
- Low Side Switches are available in Single and Dual channel configurations.

3.4 Select load type

Figure 36. Select load type

Figure 37. Resistor as load

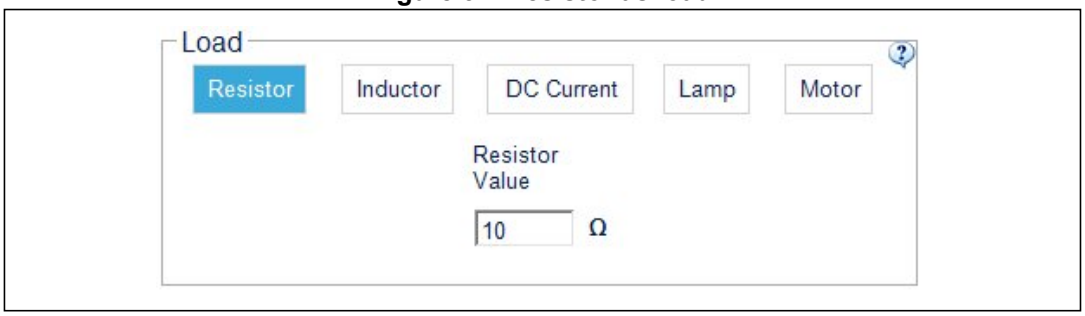


Figure 38. Inductor as load

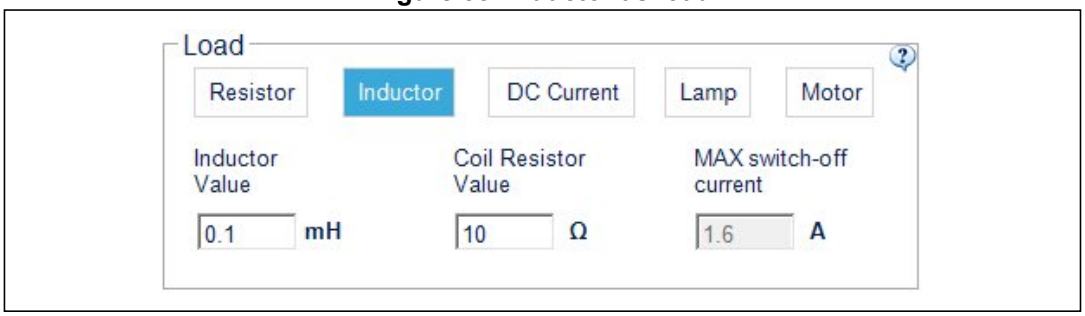


Figure 39. DC Current as load

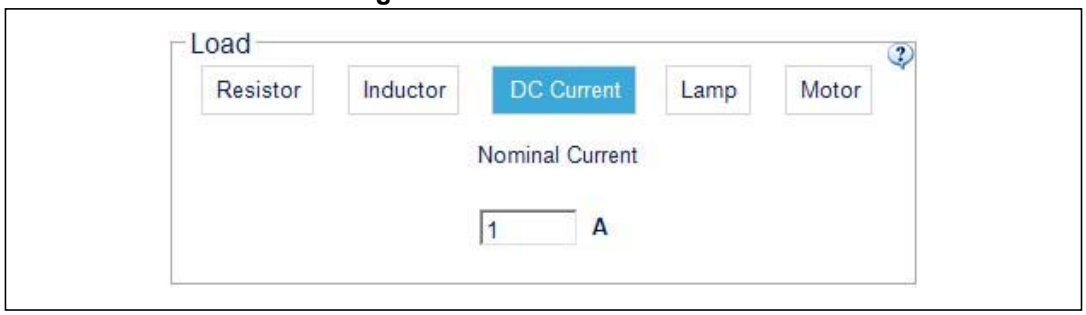


Figure 40. Lamp (by current) as load

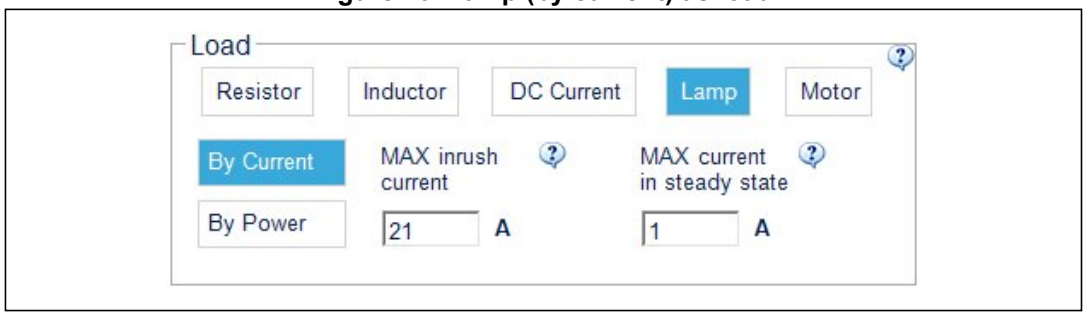


Figure 41. Lamp (by Power) as load

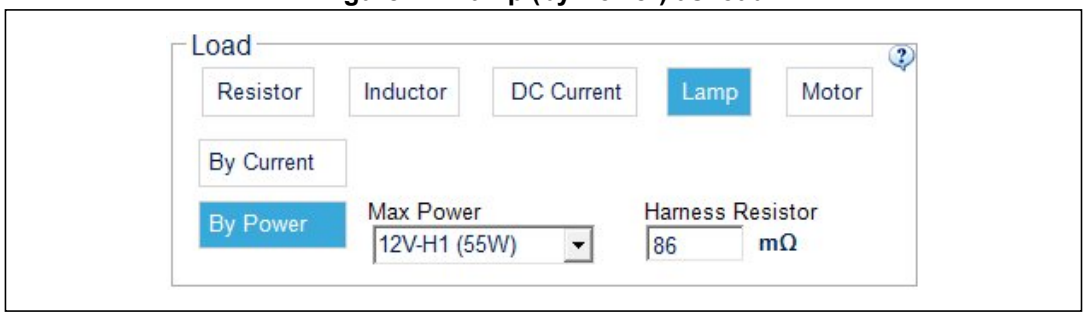
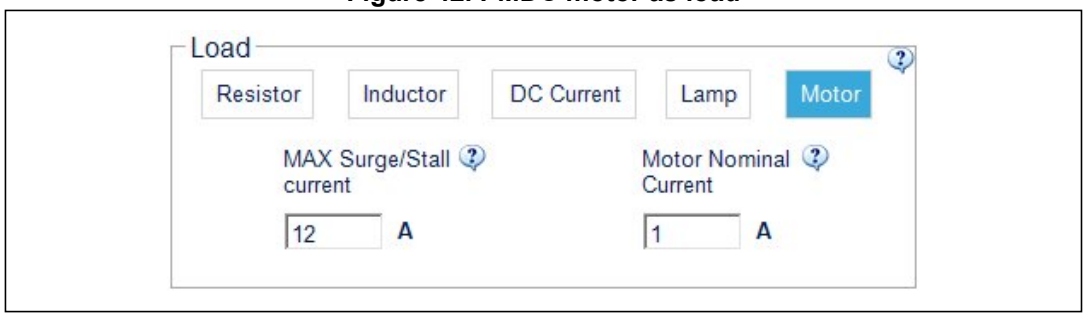


Figure 42. PMDC motor as load



Select one of the following loads:

- Resistor (*Figure 37*)
- Inductor (*Figure 38*)
- DC Current (*Figure 40*)
- Lamp (by current or by power) (*Figure 41* and *Figure 42*)
- Motor (*Figure 42*)

If "Select channels separately" is unchecked (multichannel devices only), the same load is selected for each channel.

3.5 Select source type

Figure 43. Select source type

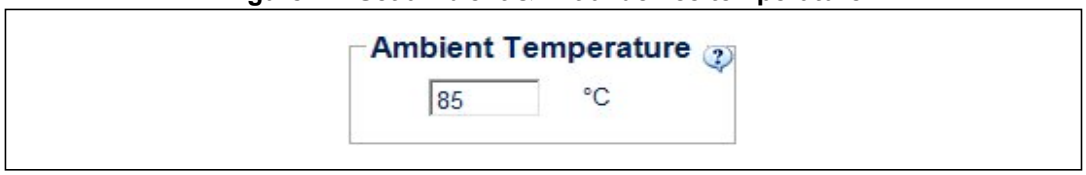
Select one of the following sources:

- DC control input
- PWM control input

If "Select channels separately" is unchecked (multichannel devices only), the same source is selected for each channel.

3.6 Set ambient & initial device temperature

Figure 44. Set ambient & initial device temperature



For standard Automotive applications 85 °C is typically used, while it is usually 105 °C for Automotive under-hood.

3.7 Select PCB type

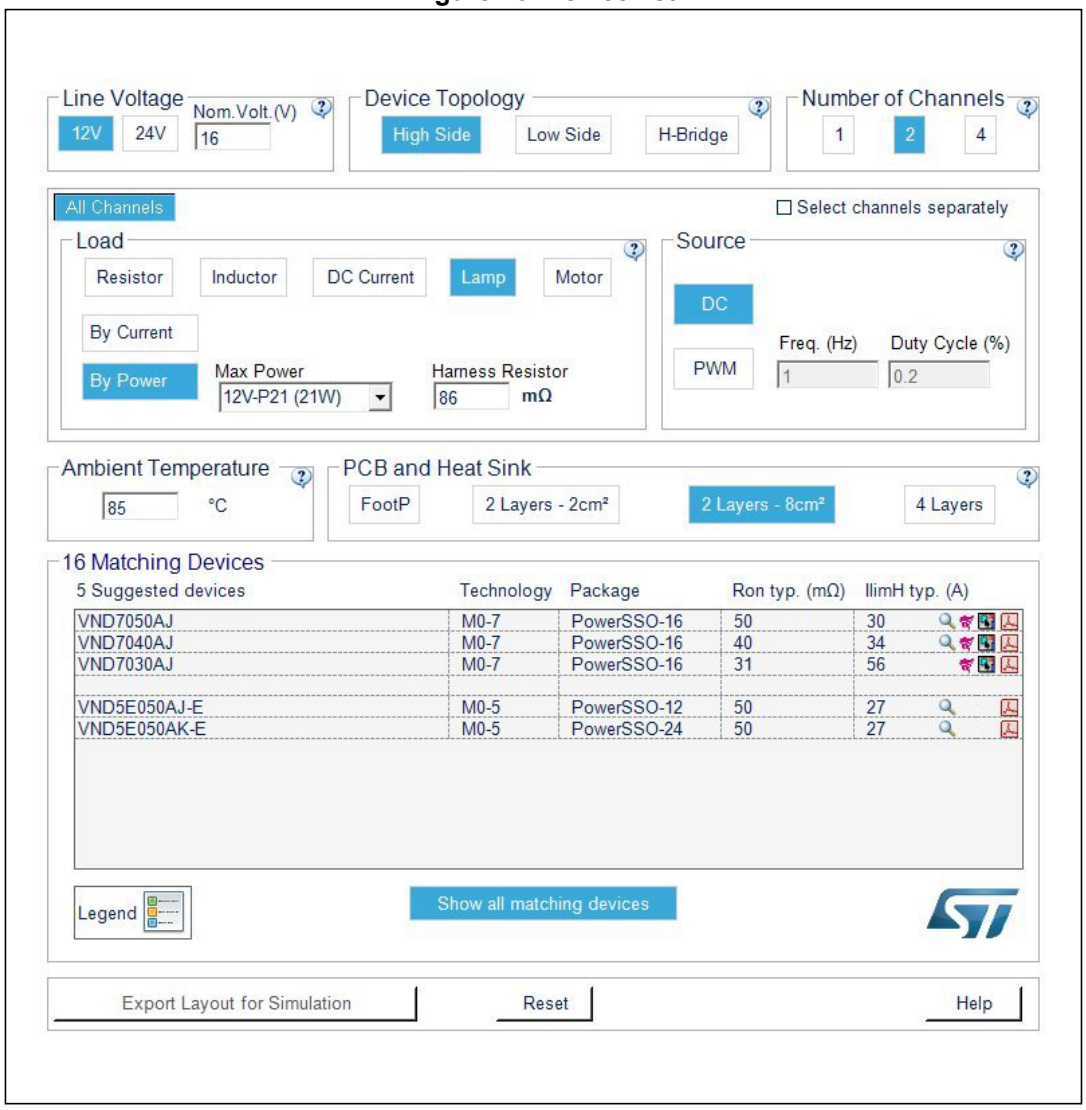
Figure 45. Select PCB type

Select the PCB type in line with the specific application:

FootPrint · 2 Layers - 2cm^2 · 2 Layers - 8cm^2 · 4 Layers

3.8 Device list

Figure 46. Device list



Select one of the devices or go back to points 1-7 and change application layout.

Devices are listed by newest technology and cheaper unit price on top.
Click "Show all matching devices" to show all devices.
If this icon appears the suggestion is to perform a detailed analysis using the ST TwisterSIM electro-themal simulator or to write an e-mail to the support center: twistersim@st.com
This icon means: Device is available for simulation.

This icon means: Device datasheet is available.

This icon means: Device orcad model is available.

4 Revision history

Table 1. Document revision history

Date	Revision	Changes
13-Mar-2015	1	Initial release.
15-Apr-2015	2	Updated *Figure 3* Deleted some contents in *Chapter 2*
29-Apr-2015	3	Updated cover page.
11-Nov-2015	4	Updated section 2.5.7 and 2.5.8 in *Section 2.6: Thermal electrical simulation setup* and *Section 2.7: Graphical simulation results*. Updated *Figure 24* and *Figure 25*. Updated *Section 2.7: Graphical simulation results*. Updated layout document.
05-Apr-2016	5	Updated *Figure 2*. Updated *Section 2.1: Device selection and parameters setting*; *Section 2.2: Source selection and setting*; *Section 2.1.2: H-Bridge Device parameters*; *Section 2.1.3: 24 V Device parameters*; *Section 2.1.4: M07 Device parameters*. Added *Figure 22*. Updated *Figure 46*.
02-Nov-2016	6	Updated *Section 2.2.1: DC voltage setting* Updated *Figure 3* Updated *Section 3.1: Select application line and nominal voltage levels* Updated *Figure 33* Updated *Section 3.2: Select device topology* Updated *Section 3.3: Select number of channels* Updated *Figure 36*, *Figure 37*, *Figure 38* Inserted new *Figure 39* Updated *Figure 40*, *Figure 41*, *Figure 42* Updated *Section 3.4: Select load type* Updated *Section 3.5: Select source type* Updated *Section 3.6: Set ambient & initial device temperature* Updated *Section 3.8: Device list* Updated *Figure 46*
13-Jul-2017	7	Updated: *Section 1* ; *Section 2.1*, *Section 2.2*, *Section 2.5.4*, *Section 3.8*. Added *Section 2.8: Monte Carlo analysis*.