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 drivers/switches and H-bridges for motor control applications.

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 online forum provides additional support to TwisterSIM users. Access to the forum is available at www.st.com/twistersim-forum.

Figure 1. TwisterSIM simulator
Contents

1 Overview ................................................................. 7

2 Electro thermal simulator ............................................. 9
   2.1 Device selection and parameters setting ...................... 9
      2.1.1 General device parameters ............................. 10
      2.1.2 H-Bridge device parameters ............................ 10
      2.1.3 24 V device parameters ................................. 10
      2.1.4 M0-7, M0-7E device parameters ......................... 10
      2.1.5 Multichannel options (2- or 4-channel devices) ........ 11
   2.2 Nominal voltage selection and setting ...................... 11
      2.2.1 DC voltage setting ........................................ 11
      2.2.2 Voltage profile setting .................................. 12
   2.3 Line and sense resistances ................................. 13
      2.3.1 Line-in resistor ........................................... 13
      2.3.2 Line-out resistors ........................................ 13
      2.3.3 Sense resistors ............................................ 14
   2.4 Control source .................................................. 15
      2.4.1 Control source type ....................................... 15
      2.4.2 H-bridge input setting ................................... 16
   2.5 Load selection .................................................. 18
      2.5.1 Resistor as load .......................................... 19
      2.5.2 RC parallel as load ....................................... 20
      2.5.3 RL series as load .......................................... 20
      2.5.4 Output current, RL load or device power loss profile as load .................................................. 21
      2.5.5 PMDC machine as load .................................... 23
      2.5.6 Lamp as load .............................................. 25
   2.6 Thermal electrical simulation setup ......................... 27
   2.7 Graphical simulation results .................................. 29
   2.8 I-T curves simulation setup .................................. 31
   2.9 Graphical simulation results (I-T curves) ................... 32
   2.10 Monte Carlo analysis ........................................... 33
      2.10.1 Monte Carlo parameters setting ......................... 35
      2.10.2 Graphical simulation results (MC analysis) ........... 37
2.11 Device-specific features ........................................... 38
  2.11.1 External LSDs ............................................. 38
  2.11.2 Microcontroller SPI ...................................... 40

3 VIPower interactive selector overview ............................... 46
  3.1 Select application line and nominal voltage levels .......... 46
  3.2 Select device topology ....................................... 46
  3.3 Select number of channels ................................... 47
  3.4 Select load type ............................................. 47
  3.5 Select source type ........................................... 49
  3.6 Set ambient & initial device temperature .................... 49
  3.7 Select PCB type ............................................. 49
  3.8 Device list .................................................. 50

4 Revision history .................................................... 52
List of tables

Table 1. Document revision history .................................................. 52
## List of figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 1</td>
<td>TwisterSIM simulator</td>
<td>1</td>
</tr>
<tr>
<td>Figure 2</td>
<td>TwisterSIM main view</td>
<td>7</td>
</tr>
<tr>
<td>Figure 3</td>
<td>Device selection and parameters setting</td>
<td>9</td>
</tr>
<tr>
<td>Figure 4</td>
<td>DC voltage setting</td>
<td>11</td>
</tr>
<tr>
<td>Figure 5</td>
<td>Voltage profile setting</td>
<td>12</td>
</tr>
<tr>
<td>Figure 6</td>
<td>Line-in resistor</td>
<td>13</td>
</tr>
<tr>
<td>Figure 7</td>
<td>Line-out resistors</td>
<td>13</td>
</tr>
<tr>
<td>Figure 8</td>
<td>Sense resistors</td>
<td>14</td>
</tr>
<tr>
<td>Figure 9</td>
<td>Control source type</td>
<td>15</td>
</tr>
<tr>
<td>Figure 10</td>
<td>PWM profile selection</td>
<td>16</td>
</tr>
<tr>
<td>Figure 11</td>
<td>H-Bridge input setting</td>
<td>17</td>
</tr>
<tr>
<td>Figure 12</td>
<td>Load selection</td>
<td>18</td>
</tr>
<tr>
<td>Figure 13</td>
<td>Resistor as load</td>
<td>19</td>
</tr>
<tr>
<td>Figure 14</td>
<td>RC parallel as load</td>
<td>20</td>
</tr>
<tr>
<td>Figure 15</td>
<td>RL series as load</td>
<td>20</td>
</tr>
<tr>
<td>Figure 16</td>
<td>Output current</td>
<td>21</td>
</tr>
<tr>
<td>Figure 17</td>
<td>RL load</td>
<td>22</td>
</tr>
<tr>
<td>Figure 18</td>
<td>Device power loss</td>
<td>22</td>
</tr>
<tr>
<td>Figure 19</td>
<td>PMDC machine as load</td>
<td>23</td>
</tr>
<tr>
<td>Figure 20</td>
<td>Lamp as load</td>
<td>25</td>
</tr>
<tr>
<td>Figure 21</td>
<td>Lamp harness resistors</td>
<td>26</td>
</tr>
<tr>
<td>Figure 22</td>
<td>Lamp harness inductor</td>
<td>26</td>
</tr>
<tr>
<td>Figure 23</td>
<td>Thermal electrical simulation setup</td>
<td>27</td>
</tr>
<tr>
<td>Figure 24</td>
<td>Simulation options</td>
<td>27</td>
</tr>
<tr>
<td>Figure 25</td>
<td>Simulation progress bar</td>
<td>28</td>
</tr>
<tr>
<td>Figure 26</td>
<td>Graphical simulation results</td>
<td>29</td>
</tr>
<tr>
<td>Figure 27</td>
<td>Create a new function to plot</td>
<td>30</td>
</tr>
<tr>
<td>Figure 28</td>
<td>I-T curves simulation</td>
<td>31</td>
</tr>
<tr>
<td>Figure 29</td>
<td>I-T curves simulation status</td>
<td>32</td>
</tr>
<tr>
<td>Figure 30</td>
<td>I-T curves simulation results</td>
<td>32</td>
</tr>
<tr>
<td>Figure 31</td>
<td>Monte Carlo analysis simulation setup</td>
<td>33</td>
</tr>
<tr>
<td>Figure 32</td>
<td>Save Monte Carlo analysis simulation data</td>
<td>34</td>
</tr>
<tr>
<td>Figure 33</td>
<td>Monte Carlo analysis wait for simulation</td>
<td>34</td>
</tr>
<tr>
<td>Figure 34</td>
<td>Monte Carlo parameters setting</td>
<td>35</td>
</tr>
<tr>
<td>Figure 35</td>
<td>Gaussian distribution diagram</td>
<td>36</td>
</tr>
<tr>
<td>Figure 36</td>
<td>Graphical simulation results (MC analysis)</td>
<td>37</td>
</tr>
<tr>
<td>Figure 37</td>
<td>External LSDs configuration</td>
<td>38</td>
</tr>
<tr>
<td>Figure 38</td>
<td>MOSFET characteristics curve</td>
<td>39</td>
</tr>
<tr>
<td>Figure 39</td>
<td>Microcontroller SPI configuration</td>
<td>40</td>
</tr>
<tr>
<td>Figure 40</td>
<td>VIPower interactive selector</td>
<td>46</td>
</tr>
<tr>
<td>Figure 41</td>
<td>Select application line and nominal voltage levels</td>
<td>46</td>
</tr>
<tr>
<td>Figure 42</td>
<td>Select device topology</td>
<td>46</td>
</tr>
<tr>
<td>Figure 43</td>
<td>Select number of channels</td>
<td>47</td>
</tr>
<tr>
<td>Figure 44</td>
<td>Select load type</td>
<td>47</td>
</tr>
<tr>
<td>Figure 45</td>
<td>Resistor as load</td>
<td>47</td>
</tr>
<tr>
<td>Figure 46</td>
<td>Inductor as load</td>
<td>47</td>
</tr>
<tr>
<td>Figure 47</td>
<td>DC current as load</td>
<td>48</td>
</tr>
<tr>
<td>Figure 48</td>
<td>Lamp (by current) as load</td>
<td>48</td>
</tr>
</tbody>
</table>
List of figures

Figure 49. Lamp (by power) as load ................................................................. 48
Figure 50. PMDC motor as load ................................................................. 48
Figure 51. Select source type ................................................................. 49
Figure 52. Set ambient & initial device temperature ................................. 49
Figure 53. Select PCB type ................................................................. 49
Figure 54. Device list ................................................................. 50
1 Overview

Main start window (Figure 2) 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 select the simulation type, customize the 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 2. TwisterSIM main view
On menu items or toolbar:
- Click "File" and select "Load System Layout & Data" or click the button, to load a complete system layout and simulation data, previously saved.
- Click "File" and select "Save System Layout & Data" or click the button, to save a complete system layout and simulation data.
- Click "File" and select "Export Layout" or click the button, to export the complete system layout in spice (orcad) netlist.
- Click "Tools" and select "VIPower Interactive Selector" or click "Get Started" button, to start VIPower Interactive Selector tool.
- Click "Help" and select "Help Topics" or click the button to view this guide and the disclaimer.
- Click "Help" and select "TwisterSIM Forum" or Click the button to visit the TwisterSIM forum page.

On system layout:
- Click on the components icons to configure and change their values.

On simulation pane:
- Select the simulation type, configure the simulation parameters (*Section 2.6: Thermal electrical simulation setup*) then click the "Simulate" button to start the 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 3) where a specific device can be selected and the related parameters can be set.

Figure 3. 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 values (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 M0-7, M0-7E 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
- V(sel2) = 5: Check to connect sel2 pin at 5V
- V(led0) = 5: Check to connect led0 pin at 5V
- V(led1) = 5: Check to connect led1 pin at 5V

Refer to the relevant datasheet for the device-specific pin subset.
2.1.5 Multichannel options (2- or 4-channel 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.

The selected device name and the 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 4) 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 4. DC voltage setting](image)

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 5) allowing to insert a custom profile by specifying time-value pairs in the corresponding fields or load values from a .txt/.csv file (time values in the 1\textsuperscript{st} column and voltage values in the 2\textsuperscript{nd}, max 150 rows).

![Figure 5. Voltage profile setting](image)

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 an increasing monotonic sequence (i.e., \( T_{n+1} > T_n \)).
- Click \( \times \) to clear table
- Click \( \mathbf{\text{Load values from file}} \) to load values from file
- Click \( \mathbf{\text{Save values to file}} \) to save values to file
- Click \( \mathbf{\text{OK}} \) to accept changes
- Click \( \mathbf{\text{Cancel}} \) to restore previous values
- Click \( \mathbf{\text{Help}} \) 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 6).

Figure 6. 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 7).

Figure 7. 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 8*).

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

*Figure 8. Sense resistors*

![Sense resistors](image)

Values outside the boundaries are automatically restored to the max/min allowed value. Accepted values are between 100 $\Omega$ and 10 k$\Omega$.

The number of enabled sense resistors is equal to the number of sense 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 9). The number of visible tabs is equal to the number of input channels of the selected device.

2.4.1 Control source type

Figure 9. 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 10) in order to insert a custom profile by specifying the 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 10. 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
Click \[\] to save values to 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 11) allows to insert a custom profile by specifying the 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 11. H-Bridge input setting

Time values must be inserted as an increasing monotonic sequence (i.e., \( T_{n+1} > T_n \)). 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 \( \times \) to reset table
Click \( \text{Load} \) to load values from file
Click \( \text{Save} \) to save values to file.


- 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 12) 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 12. Load selection

Select one of the following loads:
- Resistor (Figure 13)
- RC (Figure 14)
- RL (Figure 15)
- Output Current, RL Load or Device Power Loss Profile (Figure 16)
- PMDC Machine (Figure 19)

From the "Remove Lamp" and "Add Lamp" tabs 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 the "Resistor" button to select a resistive load and then click on the resistor symbol in the layout to change its value.

Figure 13. 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 10 mΩ and 10 MΩ 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 the "RC Load" button to select an RC parallel load and then click on the capacitor symbol in the layout to change its value.

Figure 14. 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 parallel resistance, 0 mΩ (no resistance), 10 MΩ for series 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 15. 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 available only for high-side 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 **Output current, RL load or device power loss profile as load**

Channel 1:
- Click "I/RL/W Profile" button to select an output current (I_{out}), an RL Load or a Device power loss (V_{ds} • I_{out}) 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 an output current (I_{out}) 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 16), RL Load (Figure 17) or Device power loss profile (1st channel) (Figure 18) from the pop-up menu.

---

**Figure 16. Output current**
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
Click ☐ to save values to 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 16), RL Load (Figure 17) or Power (Figure 18) 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 19. PMDC machine as load](image-url)
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·m²): 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 specific 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
- $\omega$ (rad/s): The rotational speed of the DC machine

Mechanical input (torque load):
- Hold Motor: Hold motor position ($T_L=T_e$ and $\omega=0$)
- 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., $T_{n+1}>T_n$).
- Click \(\times\) to reset table
- Click \(\rightarrow\) to load values from file
- Click \(\leftrightarrow\) to save values to file.
- Click OK to accept changes
- Click Cancel to restore previous values
- Click Help to read the help file on this topic

The PMDC machine disables the Lamp selection for the selected channel.
2.5.6 Lamp as load

Select up to 10 lamps from the "Remove Lamp" and "Add Lamp" tabs and then click on a Lamp icon in the layout to change the values.

Figure 20. 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 21. 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

Lamp harness inductor

Click on the Inductor symbol in the layout to set a wiring harness lamp inductance.

Figure 22. Lamp harness inductor

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 0 μH (no inductance) and 1 H.

- 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 the "Simulate" button to start.

**Figure 23. Thermal electrical simulation setup**

"Step Size" is automatically suggested based on the Device/Load configuration. Changing the Step Size affects the simulation speed and accuracy:
For better precision set a lower Step Size. For faster simulation set a higher Step Size.
A higher Step Size may cause convergence problems.

"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 options for more simulation parameters.

**Figure 24. Simulation options**

- Select data to plot: checked items will be available for graphical simulation results (default all).
- Check the box "Stop simulation on thermal shutdown" to interrupt the simulation when Tj is higher than thermal shutdown temperature.
A progress bar indicates the state of the simulation.

**Figure 25. 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 opens to show the simulation results.

**Figure 26. Graphical simulation results**

- Left click on the signals list checkbox to add or remove them in the plot (*Figure 26*)
- 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
- Click the "Remove Plot" and "Add Plot" tabs to show simulation results in up to 4 subplots.
- Check "Grid" to show grid lines.
- Click on "Zoom" to enable zoom for all axes.
- Click on "Zoom X" to enable zoom for horizontal axis.
- Click on "Zoom Y" to enable zoom for vertical axis.
- Click on "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 27. Create a new function to plot**

![Insert Function](image)
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.
- Click on "Plot" in the main window to show or reopen the Plot window.
- Click on "Plot" during simulation to show partial results: a notice in the plot informs about the simulation state.

2.8 I-T curves simulation setup

- Click on "I-T curves Simulation" to perform this kind of simulation for a selected device.
- Choose the “Max simulation time” and then click the "Simulate" button to Start.

The tool performs several simulations, each one with a progressive increasing value of a resistor considered as load. The resulting plot shows a curve where each point is a couple of values ("RMS value of Iout", "Time to Thermal shutdown") for increasing values of the load. In case of multi-channel devices, the load is applied only to the first channel.

Ideally, this curve should start with a simulation using a short to GND as load and end with an open load.

Actually, considering the potential huge number of simulations needed to get the I-T curve, it is requested to select an initial and a final resistance and to choose a proper sweep type, among the following three options:

- Fixed increment: load is incremented of a fixed mΩ step.
- Fixed percentage: load is incremented as a percentage of previous value.
- Auto: load is automatically incremented in order to optimize resolution and simulation speed.

"Sweep type" defines how the load resistance is incremented and this impacts the number of simulations required to draw the I-T curve. The target is to reduce the number of simulations while still keeping the required curve definition. For each option the number of simulations required to draw the plot is shown. Each simulation ends when the Tj reaches the thermal shutdown temperature or when the "Max simulation Time" is reached.

The step size is automatically calculated before each simulation starts. Changing the minimum and maximum step size affects the simulation speed and accuracy:

For better precision set a lower step size. For faster simulations set a higher step size. A higher step size may cause convergence problems.
A progress bar indicates the state of the I-T curves simulation. The process may take a lot of time.

**Figure 29. I-T curves simulation status**

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

### 2.9 Graphical simulation results (I-T curves)

When the first simulation is over, a new window opens showing the simulation results.

**Figure 30. I-T curves simulation results**

Right-click on the plot to change the titles, color, line style, signal name, unzoom, unpan, copy, save or print the image.
Click the "Zoom" button to enable zoom for all axes.
Click the "Zoom X" button to enable zoom for horizontal axis.
Click the "Zoom Y" button to enable zoom for vertical axis.
Click the "Pan" button to interactively move the plot view, left clicking on the plot.
Check "Grid" to show grid lines.
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 the last zoom.

Click the "Plot" button in the main window to show or reopen the Plot window.
Click the "Plot" button during simulation to show partial results: a notice in the plot informs about the state of simulation.
Move the "Filter Data" slider in the main window to show a filtered output waveform.

2.10 Monte Carlo analysis

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

Figure 31. Monte Carlo analysis simulation setup

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

The "Step size" is automatically selected based on the device/load configuration.
Changing the step size affects the simulation speed and accuracy:
For best precision set a lower step size. For a faster simulation, set a higher step size.
A higher step size may cause convergence problems.
After clicking "Simulate" a pop-up appears:

**Figure 32. Save Monte Carlo analysis simulation data**

- Click to change the 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 33. Monte Carlo analysis wait for simulation**

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

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

**Figure 34. 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 the thermal shutdown temperature.
Check the parameter name to perform the Monte Carlo analysis on the 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:

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

Where \( \mu \) is the mean (typical value), \( \sigma \) is the standard deviation and \( \sigma^2 \) is the variance of the distribution.
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 $\mu = (\text{max}+\text{min})/2$ and $\sigma = (\text{max}-\text{min})/10$ ensures that 99.99% of the population is in the range min-max.

A value of $\sigma$ near to zero approaches typical value, a value of $\sigma = (\text{max}-\text{min})$ approaches a uniform distribution.

Min, Max values can be changed in the parameter range. Typical value and standard deviation can be changed only for the 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

$\text{Parameter Min} + \text{Relative Value} \times (\text{Parameter Max} - \text{Parameter Min})$,

$0 \leq \text{Relative Value} \leq 1$

Click to load the table values from text or csv file

Click to save the 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.
2.10.2 Graphical simulation results (MC analysis)

- When the simulation is over, a new window opens showing the simulation results.

**Figure 36. 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.
- Click on "Zoom" to enable zoom for all axes.
- Click on "Zoom X" to enable zoom for horizontal axis.
- Click on "Zoom Y" to enable zoom for vertical axis.
- Click on "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 the "Plot" button in the main window to show or reopen Plot window.
• Click the "Plot" button during the simulation to show partial results: a notice in the plot informs about the state of simulation.

2.11 Device-specific features

• Only for H-bridge devices with integrated drivers for external power MOSFETs in low-side configuration, two external LSDs can be configured.
• Only for high-side devices with SPI interface, an external Microcontroller SPI can be configured.

2.11.1 External LSDs

Click on the MOSFETs symbols in the main window, a new frame opens where the external LSDs can be modified.

Figure 37. External LSDs configuration
Change the values manually by inserting them in the numeric boxes. Values outside the boundaries are automatically restored to the max/min allowed value.

The drain-source current $I_{DS}$ depends on the region of operation:

- In the off region ($V_{GS} < V_{th}$) the drain-source current is:
  
  $I_{DS} = 0$

- In the linear region ($0 < V_{DS} < V_{GS} - V_{th}$) the drain-source current is:
  
  $I_{DS} = K \cdot \left( ( V_{GS} - V_{th} ) \cdot V_{DS} - V_{DS}^2 / 2 \right) \cdot \left( 1 + \lambda \cdot |V_{DS}| \right)$

- In the saturated region ($0 < V_{GS} - V_{th} < V_{DS}$) the drain-source current is:
  
  $I_{DS} = \left( K / 2 \right) \cdot \left( V_{GS} - V_{th} \right)^2 \cdot \left( 1 + \lambda \cdot |V_{DS}| \right)$

Where $V_{DS}$ is the drain-source voltage, $V_{GS}$ is the gate-source voltage, $K$ is the transistor gain, $V_{th}$ is the threshold voltage and $\lambda$ is the channel modulation.

**Figure 38. MOSFET characteristics curve**

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

Click on the Microcontroller SPI symbol in the main window, a new frame opens where the external Microcontroller SPI can be modified.

Click on the Main SPI control tab for the user-friendly configuration.

Click on the Registry SPI control tab for the user hardware specific configuration.

Figure 39. Microcontroller SPI configuration

The SPI communication is based on a standard ST-SPI 16-bit interface, using CSN, SDI, SDO and SCK signal lines.

Input data are shifted into SDI, MSB first while output data are shifted out on SDO, MSB first.

Device mode:

The device can operate in different modes:

- Startup transition phase
  
  This is not an operation mode, but a transition step to reset the operation mode from the power-ON.
In this phase, neither digital supply voltage \( V_{DD} \) nor \( V_{CC} \) are available (\( V_{DD} < V_{DD\_POR\_ON} \) and \( V_{CC} < V_{USD} \)).

Characteristics
- Outputs: OFF
- SPI: inactive
- Registers: reset values
- Diagnostics: not available
- Reset bit = X

Entering conditions
- none (this is not an operating mode)

Leaving conditions
- \( V_{CC} > V_{USD} \): reset
- \( (V_{DD} > V_{DD\_POR\_ON}) \) and \( (V_{CC} < V_{USD}) \): Fail Safe

• Reset mode
Characteristics
- Outputs: according to INx
- SPI: inactive
- Registers: reset values
- Diagnostics: not available
- Reset bit = X

Entering conditions
- Startup mode: \( V_{CC} > V_{USD} \)
- Sleep 1: INx Low to High
- Any other mode: \( V_{DD} < V_{DD\_POR\_OFF} \)

Leaving conditions
- All INx low: sleep 1
- \( V_{DD} > V_{DD\_POR\_ON} \): Fail Safe

• Fail safe mode
Characteristics
- Outputs: according to INx
- SPI: active
- Registers: read/write possible, cleared if entered after SW reset
- Diagnostics: SPI possible, CurrentSense diagnostic is not possible
- Reset bit = 1 if entered after SW reset or POR, else Reset bit = 0

Entering conditions
- Reset or sleep 1: \( V_{DD} > V_{DD\_POR\_ON} \)
Standby or sleep 2: CSN low for $t > t_{\text{stdby\_out}}$
Normal: $EN = 0$ or WDTB toggling timeout or SW-reset

Leaving conditions
- $V_{DD} < V_{DD\_POR\_OFF}$: reset
- SPI sequence 1. UNLOCK = 1, 2. STBY = 0 and EN = 1: normal
- SPI sequence 1. UNLOCK = 1, 2. STBY = 1 and EN = 0: Standby

- Normal mode
  Characteristics
  - Outputs: according to SPI register settings and/or INx
  - SPI: active
  - Registers: read/write is possible
  - Diagnostics: SPI and CurrentSense diagnostic possible
  - Regular toggling of WDTB is necessary within timeout period $t_{WDTB}$
  - Reset bit = 0

Entering conditions
- Fail Safe: SPI sequence 1. UNLOCK = 1, 2. STBY = 0 and EN = 1

Leaving conditions
- $V_{DD} < V_{DD\_POR\_OFF}$: reset
- SPI sequence 1. UNLOCK = 1, 2. STBY = 1 and EN = 0: Standby
- EN = 0 or WDTB time out or SW reset: Fail-Safe

- Standby mode
  Characteristics
  - Outputs: according to INx
  - SPI: inactive
  - Registers: frozen
  - Diagnostics: not available
  - Low supply current from $V_{DD}$
  - CSN: High
  - Reset bit = 0
Entering conditions
- Normal: SPI sequence 1. UNLOCK = 1, 2. STBY = 1 and EN = 0
- Fail Safe: SPI sequence 1. UNLOCK = 1, 2. STBY = 1 and EN = 0
- Sleep 2: INx low to high

Leaving conditions
- \( V_{DD} < V_{DD\_POR\_OFF} \): Reset
- CSN low for \( t > t_{stdby\_out} \): Fail-Safe
- All INx low: sleep 2

• Sleep mode 1
Characteristics
- Outputs: OFF
- SPI: inactive
- Registers: reset values
- Diagnostics: not available
- Low supply current from \( V_{DD} \) and \( V_{CC} \)
- Reset bit = X

Entering conditions
- Reset: all INx = 0

Leaving conditions
- \( V_{DD} > V_{DD\_POR\_ON} \): Fail Safe
- INx low to high: reset

• Sleep mode 2
Characteristics
- Outputs: OFF
- SPI: inactive
- Registers: frozen
- Diagnostics: not available
- Low supply current from \( V_{DD} \) and \( V_{CC} \)
- CSN: High
- Reset bit = 0
Entering conditions
- Standby: all INx = 0

Leaving conditions
- $V_{DD} > V_{DD\_POR\_OFF}$: reset
- CSN low for $t > t_{stdby\_out}$: Fail-Safe
- INx low to high: Standby

- Battery undervoltage mode
  This is not an operation mode but a transition step, where power supply voltage is $V_{CC} < V_{USD}$.
  If the battery supply voltage $V_{CC}$ falls below the undervoltage shutdown threshold ($V_{CC} < V_{USD}$) the device enters Battery undervoltage mode.
  The CurrentSense signal is not available.
  The output stages are off regardless of the SPI status or INx.

The Reset mode, the Fail Safe mode and the Sleep mode 1 are combined into the Limp home mode. In this mode the chip is able to operate without the connection to the SPI. All transitions between the states in limp home mode are driven by $V_{DD}$ and INx. The outputs are controlled by the direct inputs INx.

Thermal threshold:
Set the temperature threshold warning from the pop-up menu in the Main SPI control tab or from Control register bits 1,2 in the Registry SPI control tab.

SPI output control:
The SPI Output Control register controls the output drivers in Normal Mode.
1: The corresponding output is enabled
0: The corresponding output is disabled
Set values from the ChX On/Off buttons in the Main SPI control tab or from SOCR bits 0...3 in the Registry SPI control tab.

INx:
In Fail safe, Standby and Reset modes, external control signals INx turn ON/OFF the outputs by applying high/low logic levels to the dedicated pin.
In normal mode, control the outputs if the corresponding bit in the Direct Input Enable Control Register (DIENCR) is at logic high level.
Set values from the On/Off buttons in the Main SPI control tab.

CurrentSense Multiplexer:
Check "CsMUX enable" in the Main SPI control tab or set the CurrentSense Multiplexer Control Register (CSMUXCR) bit 3 in the Registry SPI control tab to enable the CurrentSense output.
Select the CurrentSense mux Channel from CH0...CH3 buttons in the Main SPI control tab.
or from CSMUXCR bits 0...2 in Registry SPI control tab.

**Latch Off:**

Set Channel Latch OFF Timer Control Registers (ch0, ch1) and (ch2, ch3)
The CHLOFFTCR0,1 and CHLOFFTCR2,3 configure the blanking time duration in case of
power limitation or overtemperature for the corresponding output.
In Normal Mode, the output behavior in case of power limitation or thermal shutdown is
programmable, as latch-off, time limited auto-restart ($t_{blanking}$). The default mode is the
latch-off mode.
In latched off-state the fault has to be cleared to re-enable the output channel after an
overtemperature or power limitation event through a new value written through SPI
command at CHLOFFTCRx register.
In fail-safe state, the device operates in unlimited auto-restart mode.
Set values from the trackbars in the Main SPI control tab or from CHLOFFTCR0,1 and
CHLOFFTCR2,3 in the Registry SPI control tab.

**Mode:**

Set the Channel Control Register (CCR)
The CCR selects the BULB or LED mode for the corresponding output
1: Led mode selected for OUTPUTX
0: Bulb mode selected for OUTPUTX
Set values from the Led/Bulb buttons in Main SPI control tab or from CCR bits 0,1 in
Registry SPI control tab.

**Slew Rate:**

Set the Fast Switching Configuration Register (FASTSWCR)
The FASTSWCR allows to configure each channel in normal or fast switching mode
1: Fast Switch
0: Normal Switch
Set values from the Fast/Normal buttons in Main SPI control tab or from FASTSWCR bits
0...3 in Registry SPI control tab.

**Open-load OFF-state:**

Set the Open-load OFF-State Control Register (OLOFFCR)
The OLOFFCR enables an internal pull-up current generator to distinguish between the
open-load OFF-state fault and the output shorted to $V_{CC}$ fault.
1: Pull-up current generator enabled for OUTPUTX
1: Pull-up current generator disabled for OUTPUTX
Set values from the checkboxes in Main SPI control tab or from OLOFFCR bits 0...3 in
Registry SPI control tab.

Changes on the Main SPI control tab are reflected on the Registry SPI control tab and vice
versa.
Values can be changed during the simulation and the device state is reflected on the GUI.
Refer to the relevant datasheet for device-specific parameter ranges.
- Click **OK** to accept changes.
- Click **Cancel** to restore previous values.
- Click **Help** to read the help file on this topic.
3 VIPower interactive selector overview

Figure 40. VIPower interactive selector

3.1 Select application line and nominal voltage levels

Figure 41. 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 trucks 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 42. 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 motor driving, either the high-side or the 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 43. 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 44. Select load type

Figure 45. Resistor as load

Figure 46. Inductor as load
Select one of the following loads:
- Resistor (Figure 45)
- Inductor (Figure 46)
- DC Current (Figure 48)
- Lamp (by current or by power) (Figure 49 and Figure 50)
- Motor (Figure 50)
If "Select channels separately" is unchecked (multichannel devices only), the same load is selected for each channel.

3.5 Select source type

Figure 51. 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 52. 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 53. Select PCB type

Select the PCB type in line with the specific application:
FootPrint · 2 Layers - 2cm² · 2 Layers - 8cm² · 4 Layers
3.8 Device list

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

Devices are listed by the newest technology and the 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-thermal 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 ![icon] means: Device orcad model is available.

- Click "Export Layout for Simulation" to verify the selected layout in TwisterSIM for the available devices.
- Click "Reset" to restart VIPower Interactive Selection.
- Click "Help" to read the help file on this topic.
# Revision history

## Table 1. Document revision history

<table>
<thead>
<tr>
<th>Date</th>
<th>Revision</th>
<th>Changes</th>
</tr>
</thead>
<tbody>
<tr>
<td>13-Mar-2015</td>
<td>1</td>
<td>Initial release.</td>
</tr>
<tr>
<td>15-Apr-2015</td>
<td>2</td>
<td>Updated Figure 4. Deleted some contents in Chapter 2.</td>
</tr>
<tr>
<td>29-Apr-2015</td>
<td>3</td>
<td>Updated cover page.</td>
</tr>
<tr>
<td>11-Nov-2015</td>
<td>4</td>
<td>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 26 and Figure 27. Updated Section 2.7: Graphical simulation results. Updated layout document.</td>
</tr>
<tr>
<td>05-Apr-2016</td>
<td>5</td>
<td>Updated Figure 3. 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: M0-7, M0-7E device parameters. Added Figure 24. Updated Figure 54.</td>
</tr>
<tr>
<td>02-Nov-2016</td>
<td>6</td>
<td>Updated Section 2.2.1: DC voltage setting Updated Figure 4 Updated Section 3.1: Select application line and nominal voltage levels Updated Figure 41 Updated Section 3.2: Select device topology Updated Section 3.3: Select number of channels Updated Figure 44, Figure 45, Figure 46 Inserted new Figure 47 Updated Figure 48, Figure 49, Figure 50 Updated Section 3.4: Select load type Updated Section 3.5: Select source type Updated Section 3.6: Set ambient &amp; initial device temperature Updated Section 3.8: Device list Updated Figure 54</td>
</tr>
<tr>
<td>13-Jul-2017</td>
<td>7</td>
<td>Updated: Section 1, Section 2.1, Section 2.2, Section 2.5.4, Section 3.8. Added Section 2.10: Monte Carlo analysis.</td>
</tr>
<tr>
<td>08-Nov-2019</td>
<td>8</td>
<td>Updated the entire document (content and graphics).</td>
</tr>
</tbody>
</table>