



Vincotech

Table of Contents

VincotechISE User Manual

Integrated Simulation Environment for
Vincotech Power Modules





Table of Contents

1	Abstract	4
2	Introduction	4
3	Installing Vincotech ISE2	4
4	Launching Vincotech ISE2	4
4.1	Updating	6
4.2	Version Information	6
4.3	Support	7
5	Working with Vincotech ISE	8
6	Simulation Tools	10
6.1	<i>flowSOL</i>	10
6.2	<i>flowSIM</i>	19
6.3	<i>flowSEL</i>	27



Vincotech

Date	Revision Level	Description	Page Number(s)
Aug. 2008	1	VincotechISE - first release	17
Apr. 2014	2	VincotechISE2, - second release	31
May 2015	3	VicotechISE – adjusted for ISE versions beginning with 3.xx	30

1 Abstract

Vincotech ISE is a set of integrated simulation tools designed to help you select power modules for various applications. All power loss and temperature calculations are based on actual measurements taken of each module.

2 Introduction

Vincotech ISE features the tools for *flowSOL*, *flowSIM* and *flowSEL*. Besides two other tools, *flowMIS* and the new thermal simulator *flowTHERM*, are available; calculations can be provided on special request. These software components are here to support your efforts to develop electronic applications and find the power module best suited for the task at hand.

3 Installing Vincotech ISE2

Vincotech ISE is based on the labVIEW runtime engine, so you'll need it to launch our simulation tool.

Go to www.vincotech.com to get the right version. Once it is installed, download Vincotech ISE2 from our website at <http://www.vincotech.com/en/support/simulation-software.html>. Save it to the simulation directory of your choice.

4 Launching Vincotech ISE2

Only the executable file is downloaded, so when you first launch the tool a Download/Update window appears showing the latest data files. They contain the modules' parametrical data and other information needed to run the software. Click the download button to retrieve these data.

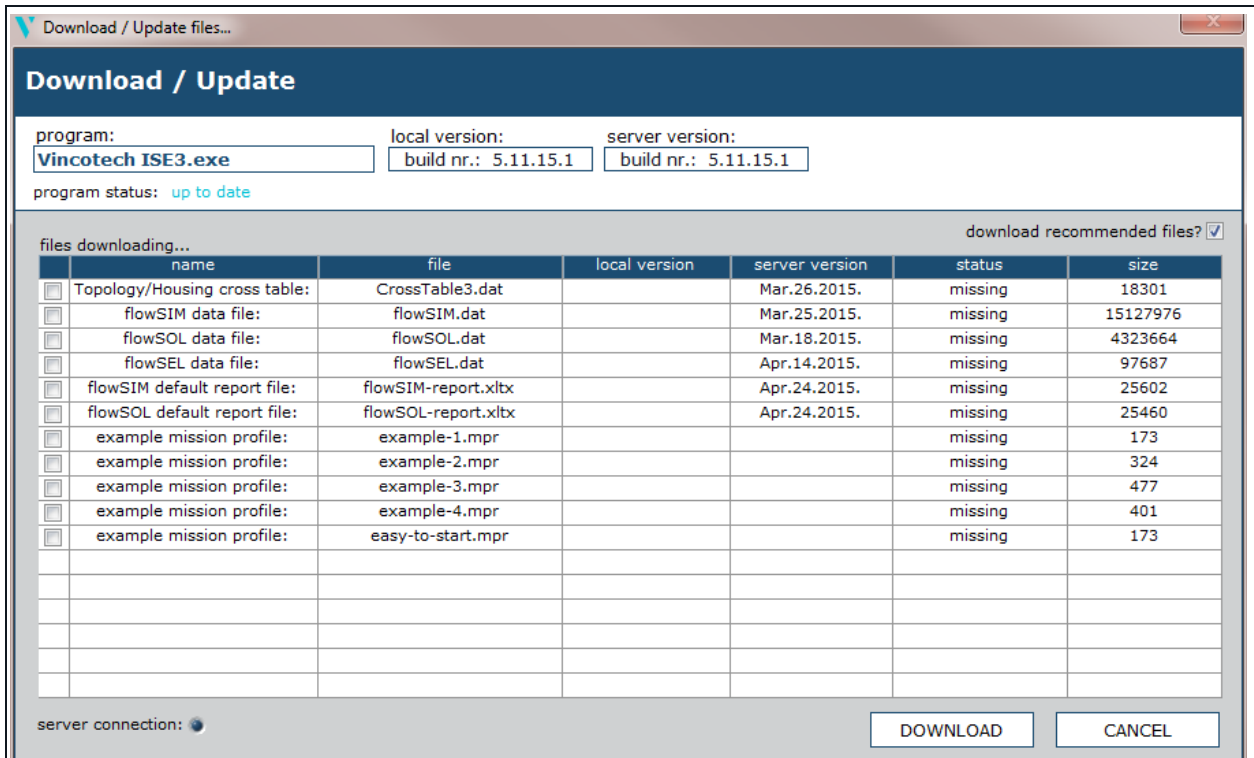


Figure 1: Download/Update window

Once all files are up to date, click the finish button to close the window. All data files are stored in a folder named 'data' generated in the chosen simulation directory. The program stops if the download fails, in which case you'll have to restart it and try again.

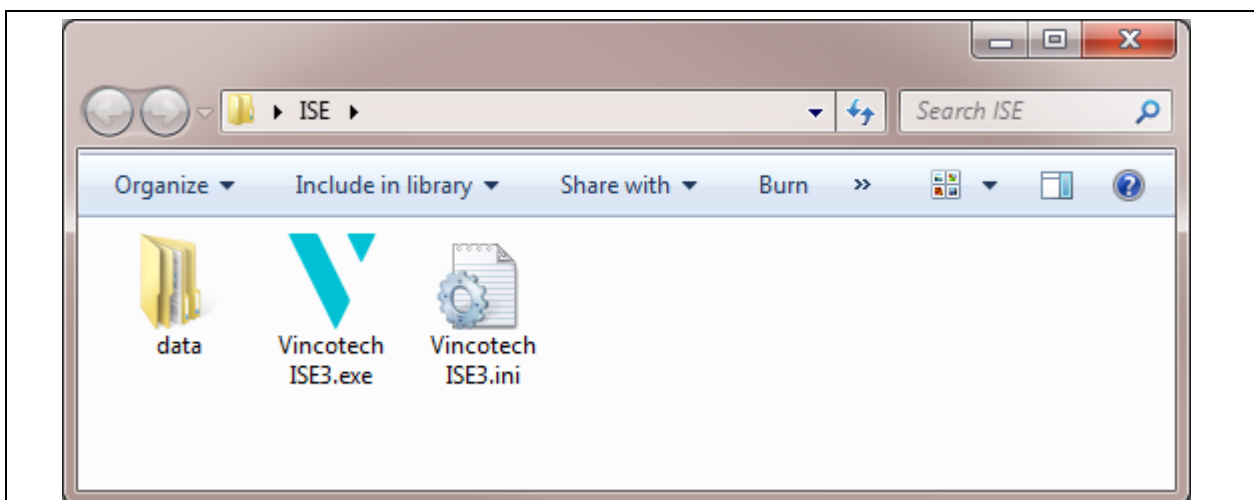


Figure 2: Root folder content

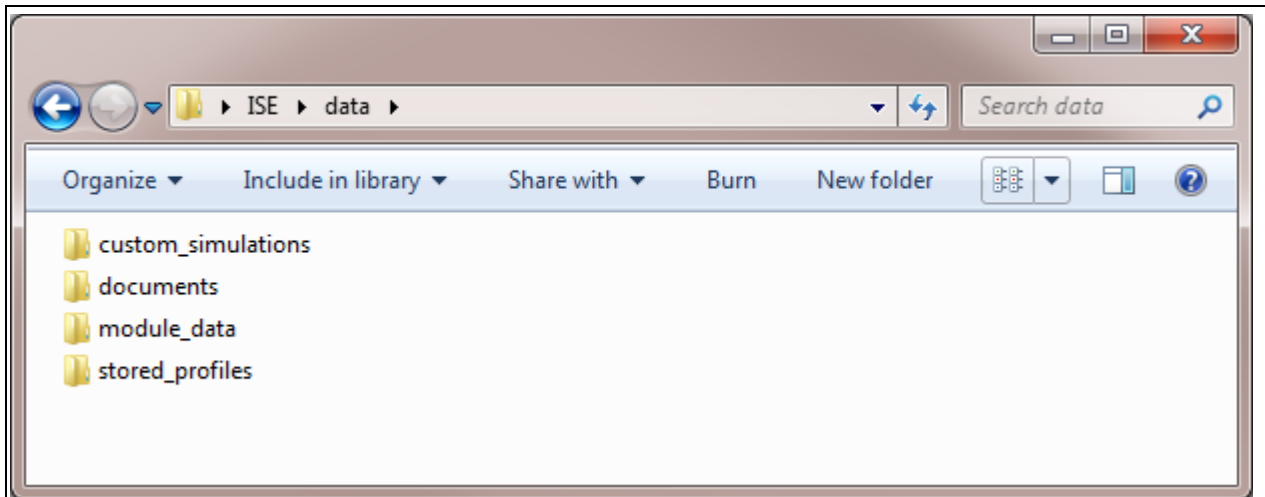


Figure 3: Data folder content

4.1 Updating

The program starts checking the revision history of files on the server at every start of VincotechISE. If more recent versions of files are found, they appear in the Download/Update window. If so, click the download button to load all files to the 'data' folder. There is no need to delete old files manually as the program uses only the latest versions of data files.

If an updated version of Vincotech ISE2 is available, you will have to restart the program after downloading the new files.

4.2 Version Information

Click 'Version information' in the help menu to learn more about every software component and data file.

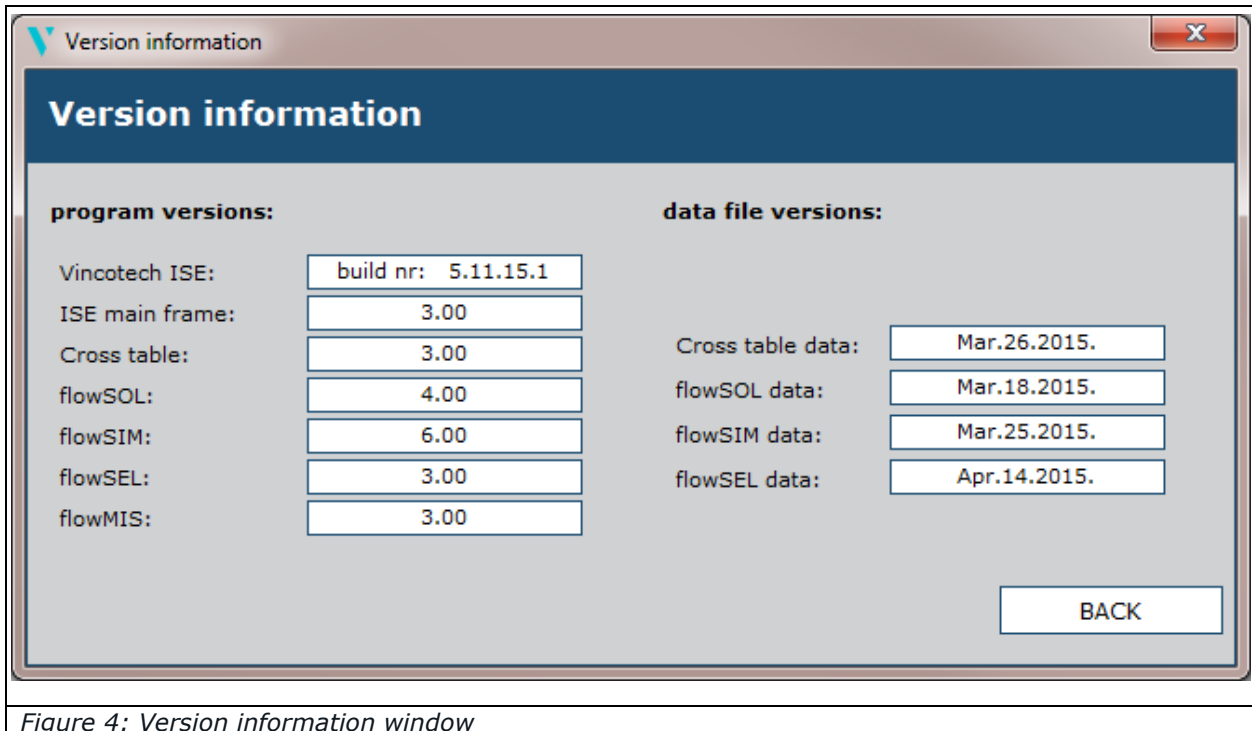


Figure 4: Version information window

4.3 Support

If you need any help, go to the contact menu and click the support button to see a list of Vincotech’s regional field application engineers and developers. Links to Vincotech’s latest application notes, reference designs and technical papers are also posted here.

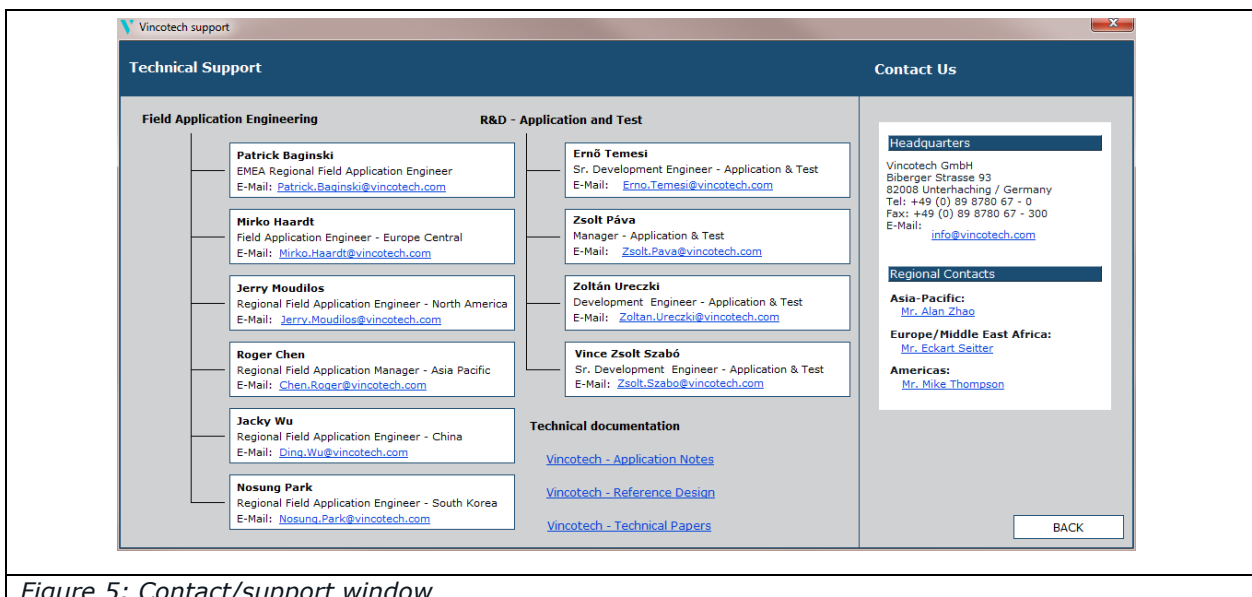
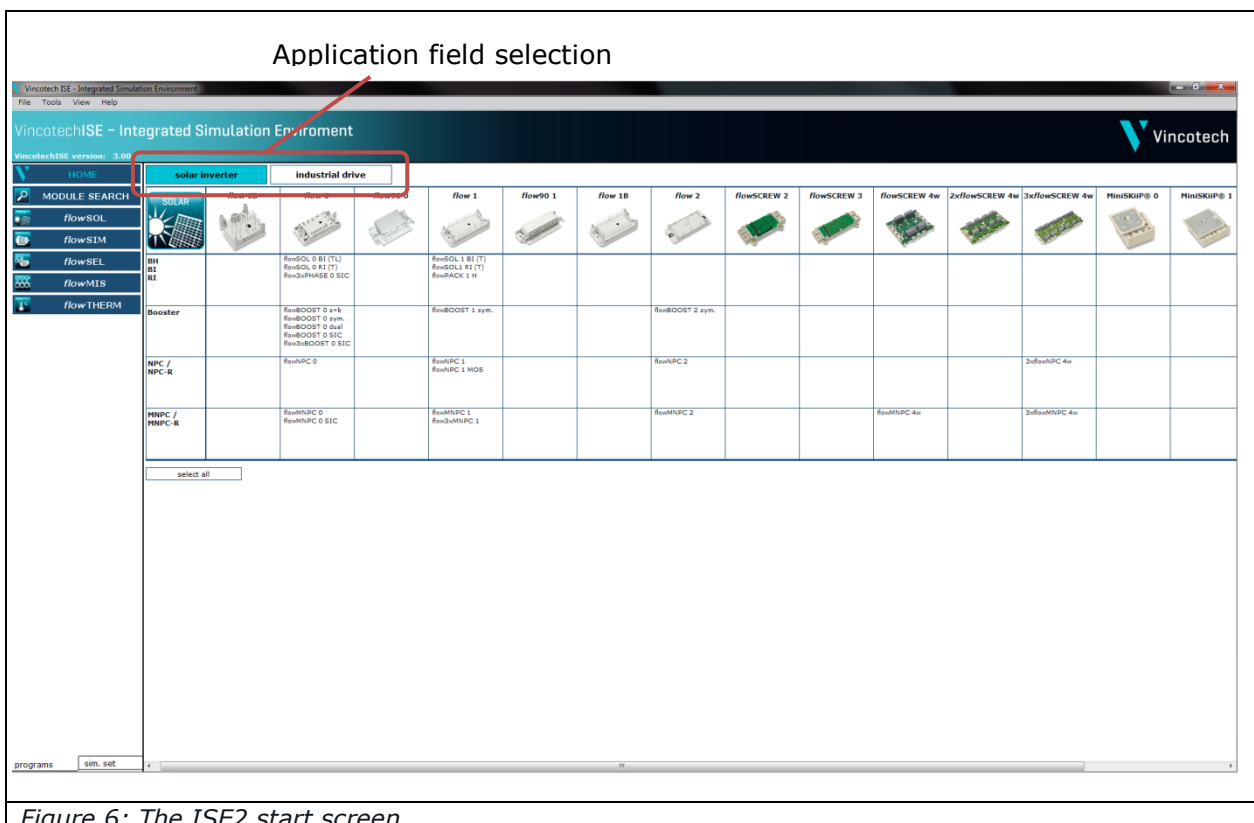


Figure 5: Contact/support window

5 Working with Vincotech ISE

Once all files are up to date, Vincotech ISE is ready to simulate modules. There are two ways to get the simulation of the power module of choice started; choose the preferred product family via a cross-reference table or select a concrete power module by typing the name in the module search.

If only a preferred topology is specified at the start of the simulation tool, the table cross-referencing *flow* and MiniSKiiP power module housings with topologies helps you to find the right product family. This table appears directly after launching Vincotech ISE. Select either 'solar inverter' or 'industrial drive' at the top of the table to change the application field. The selection of topologies will change accordingly.



Now select your preferred topology by clicking it in the column of your chosen housing. Then click the *flowSOL* button for solar inverters or the *flowSIM* button for industrial drives to start the simulation. Use the control key if you wish to select multiple modules with the same

topology. If you don't select a topology now, you will be asked to do so when the selected simulation tool launches.

In case of an already preselected Vincotech power module, you can easily find this module by typing the product name in the search engine. Just click on the button module search to open the search function.

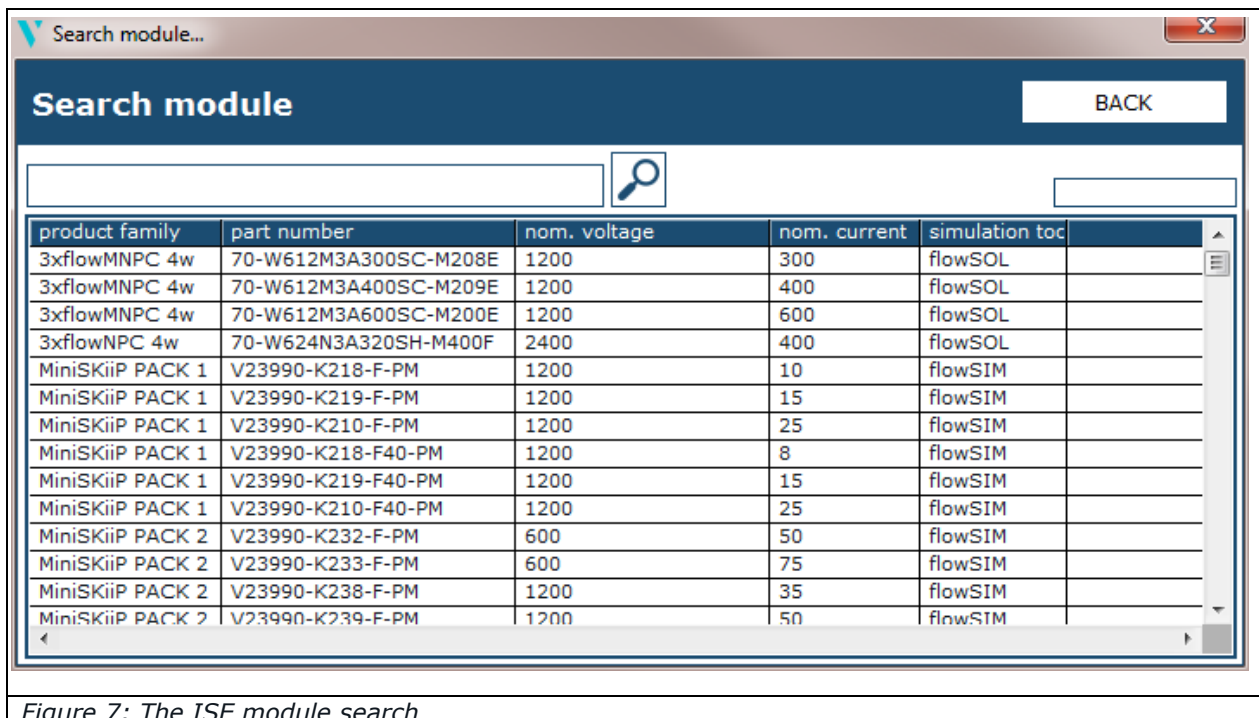







Figure 7: The ISE module search

Type the name (or only a part of it) of the selected power module in the search bar and click on the magnifier button. Afterwards, all suitable power modules will be listed here. Just double click on the preferred one and the simulation starts at once. In some cases, topologies respectively power modules can be simulated in *flowSOL* and *flowSIM*, then please check in column "simulation tool" which tool fits best to your application and select the module accordingly.

If you wish to open and continue a stored simulation, the 'custom simulations' window can be opened by clicking on 'sim. set' at the lower left corner of the screen. It serves to open former simulations and to save the currently configured simulation. Click the plus button and name the current simulation. It is stored automatically in the given simulation tool folder.

6 Simulation Tools

All other Vincotech ISE2 components are launched by clicking the appropriate buttons in the top row of the start screen:

 flowSOL	Simulation program for solar inverter modules
 flowSIM	Simulation program for industrial drive modules
 flowSEL	Selector tool for drive modules
 MODULE SEARCH	Opens the search function
 HOME	Takes you back to the start screen

6.1 *flowSOL*

This tool simulates Vincotech power modules designed for solar applications. The user interface provides several windows and entry masks where you can adjust all the key parameters. The window consists of two panels. The top panel contains all modules' parameters and the simulation section; the bottom panel shows graphs with information on voltage, current, temperature and losses.

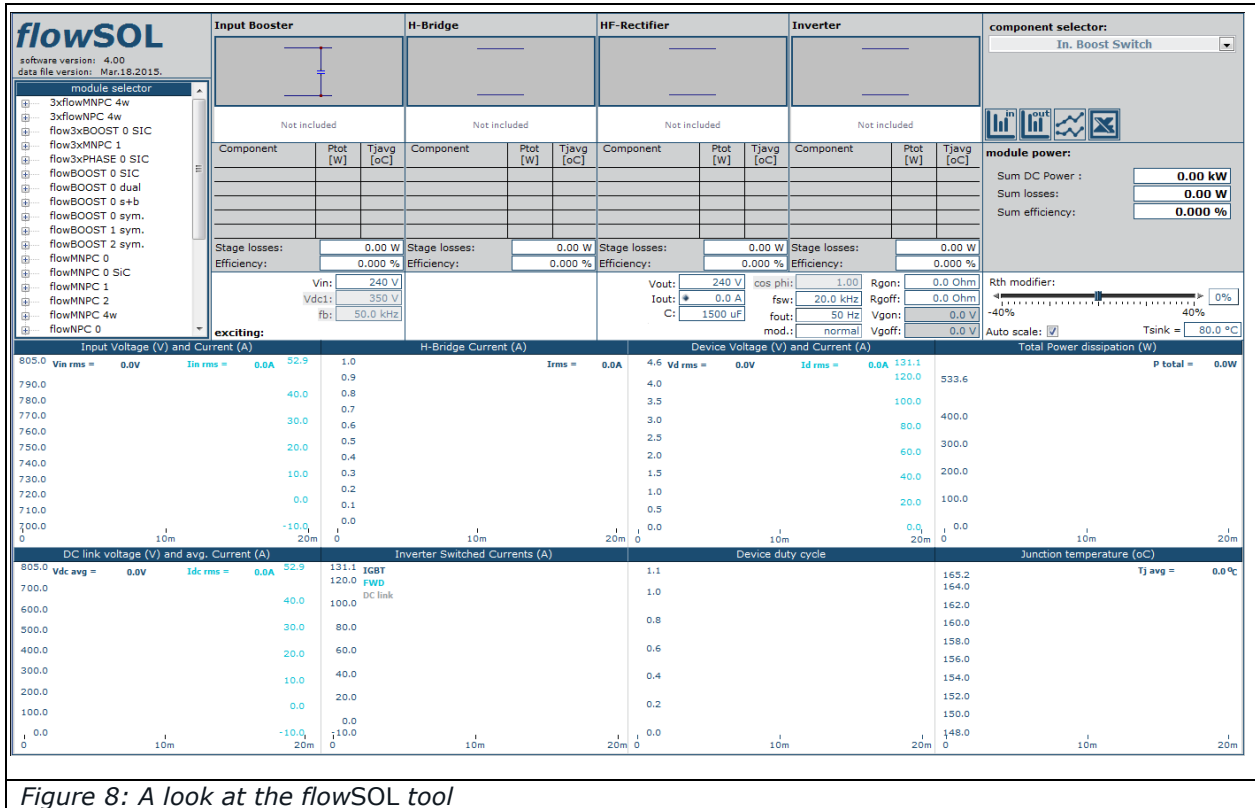


Figure 8: A look at the flowSQL tool

You must choose a specific module in the module selector window. It shows all available modules for the topologies selected on the Vincotech ISE2 start screen.

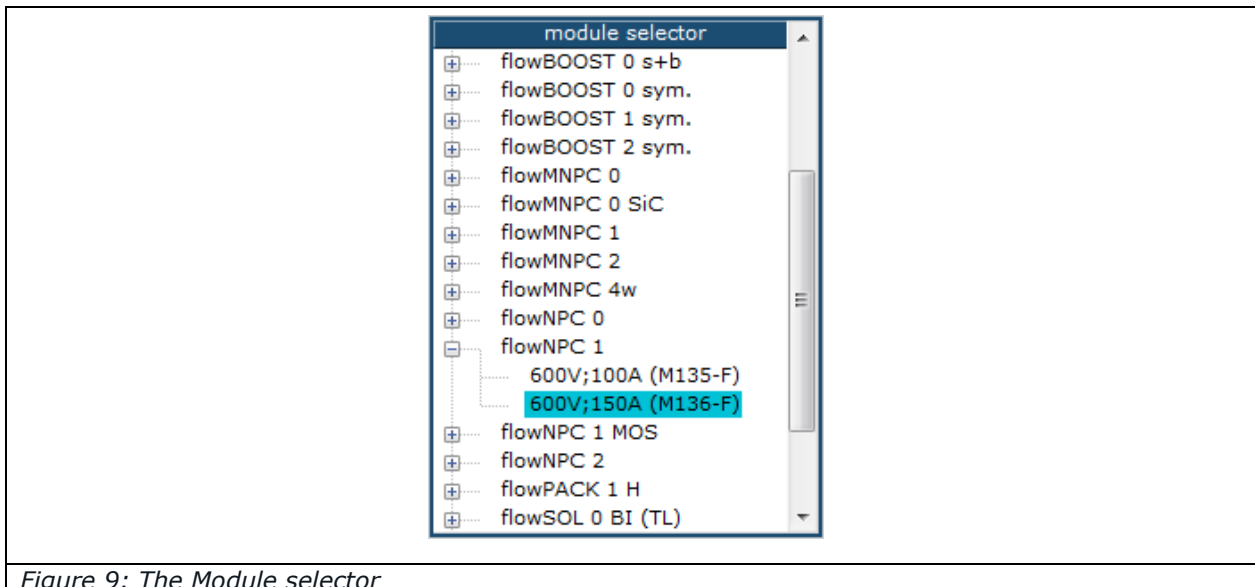


Figure 9: The Module selector

The simulation starts as soon as you choose a module, using the preset parameters as the basis. The top row of the upper panel is divided by function into four boxes called Input Booster, H-Bridge, HF-Rectifier and Inverter. These boxes are disabled (empty and framed in dark blue) or enabled (filled with a circuit diagram and framed in polar blue) in accordance with your chosen topology. The selected power module's full name is also indicated. Below these boxes you'll see listed the circuit's components and information about their losses and average temperature.

A right-click on an enabled box opens the module selector. It lists all power modules that you can substitute to perform this function. This way of choosing a replacement has the benefit that your parameters are retained. If instead you select a new module in the module selector on the left, all parameters are reset to their defaults.

Right-clicking a disabled function box also opens the module selector. In this case, a combination of different power modules, say a booster and an inverter, can be simulated.

The far right column indicates DC power, overall losses and the module's efficiency. The stated values always refer to both power modules when simulating a combination of modules.

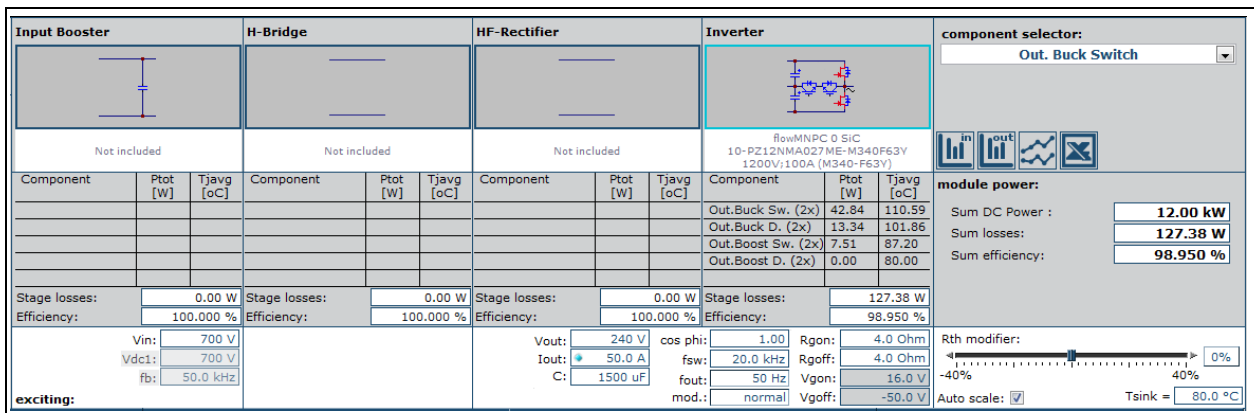
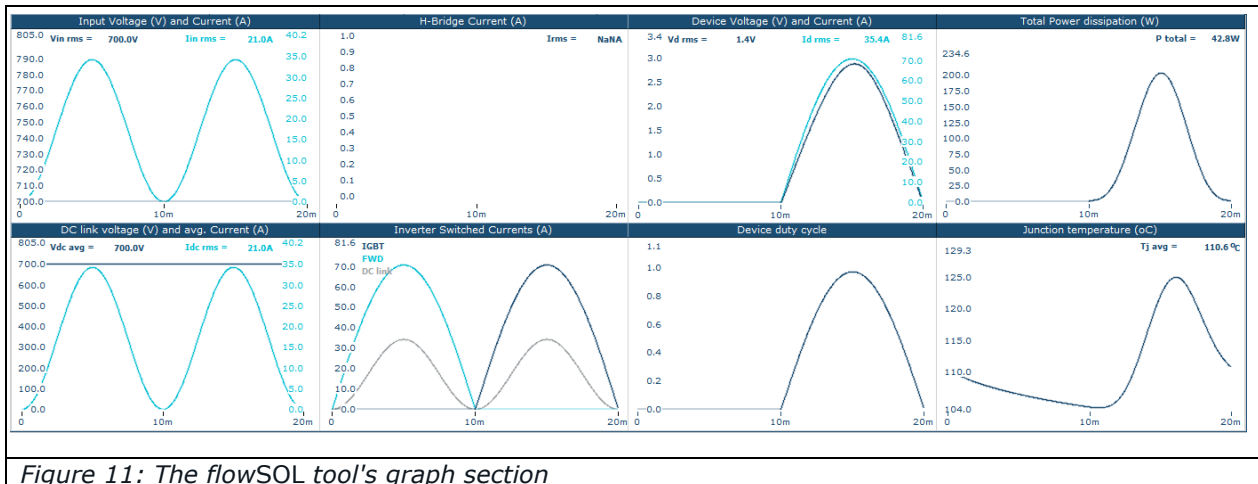


Figure 10: The flowSQL tool's function boxes and entry mask

Now you can fine-tune input/output voltage, output current, switching frequency and all other parameters for the selected topology. Depending on the enabled function box and power module, you can adjust:

- V_{in} – input voltage in [V]
- V_{dc1} – booster output voltage in [V]
- f_b – booster switching frequency in [Hz]
- V_{out} – phase voltage in [V]
Line-to-ground RMS value
- I_{out} – RMS output current in [A]
A green light indicates that negative currents may also be simulated.
- C – DC link capacitance in [μ F]
- $\cos \phi$ – efficiency factor $\cos \phi$, range 0 to 1
- f_{sw} – switching frequency in [Hz]
Configuration-dependent: H-bridge or inverter
- f_{out} – output/grid frequency in [Hz]
Available presets: 50 Hz and 60 Hz
- R_{gon} – switch-on gate resistor in [Ω]
- R_{goff} – switch-off gate resistor in [Ω]
Value must be equal to or lower than switch-on gate resistor R_{gon} .
- T_{sink} – heat sink temperature in [$^{\circ}$ C]
- Inverter modulation
Configuration-dependent presets: normal, unipolar, bipolar, 3rd harmonic
- R_{th} mod.
This slider allows to adjust the preset R_{th} , if in the application a better or worse thermal resistance than in the datasheet stated is expected
- V_{gon} – positive gate driver voltage in [V]. This preset is nonadjustable.
- V_{goff} – negative gate driver voltage in [V]. This preset is nonadjustable.

The graphs in the *flowSOL* screen's lower panel are updated after every parameter change. These graphs are windows into the simulation affording you views of input voltage curves, input current, DC link values and the like to show how individual components respond to the given parameters. Select the component you wish to analyze in the component selector box at the right of the top panel.



To gain deeper insight into the chosen component's input and output waveforms, left-click the three buttons in the simulation section to see detailed graphs indicating:



Input and output waveforms



Static and dynamic losses and temperature graphs



Static and switching characteristics



Generates an Excel report file



Detailed input and output voltage and current waveforms
(independent of the selected component)

Input and H-bridge

DC link and rectifier

Output

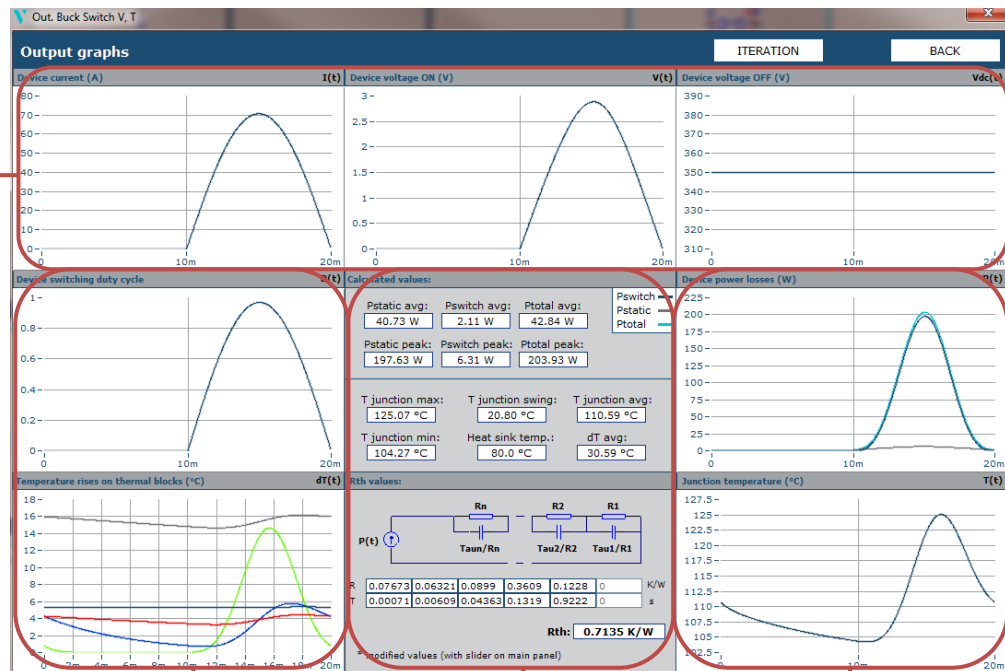


Figure 12: Detailed input and output waveforms in the flowSOL tool



Detailed static and dynamic losses and temperature graphs for the selected component

Current, voltage



Duty cycle, temperature rise

Power losses, junction temperature, thermal resistance

Power losses, junction temperature

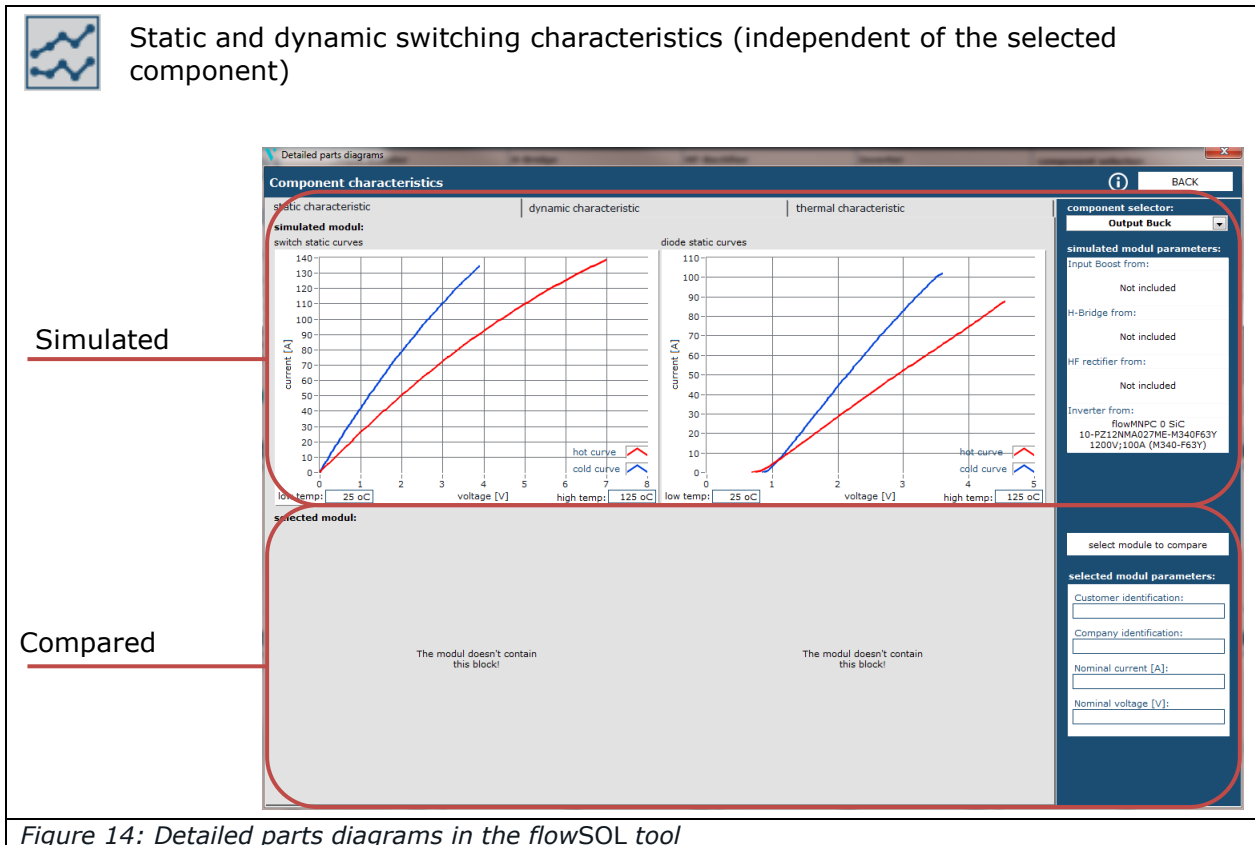
Figure 13: Detailed static and dynamic losses and temperature graphs in the flowSOL tool

The 'Detailed parts diagrams' window serves to analyze static and dynamic switching characteristics and thermal impedance. Informative graphs are shown on various pages depending on the selected module's configuration as an input booster, H-bridge, HF rectifier or inverter.

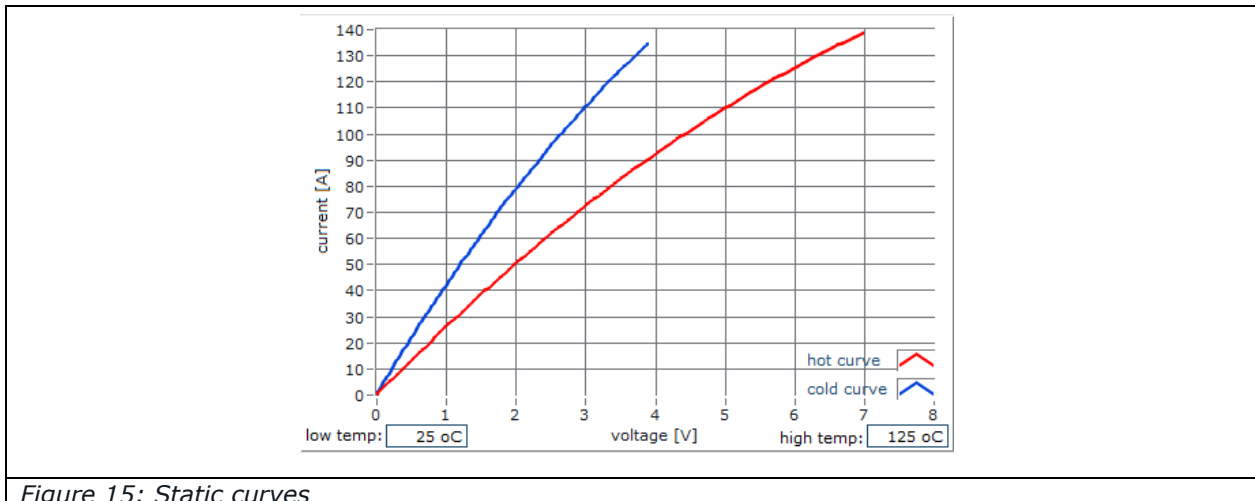
You can also select a second power module to compare two options directly. To do this, open the module selector by clicking the 'select module to compare' button on the right. A second module is then simulated using the same set of parameters as for the first.



Static and dynamic switching characteristics (independent of the selected component)

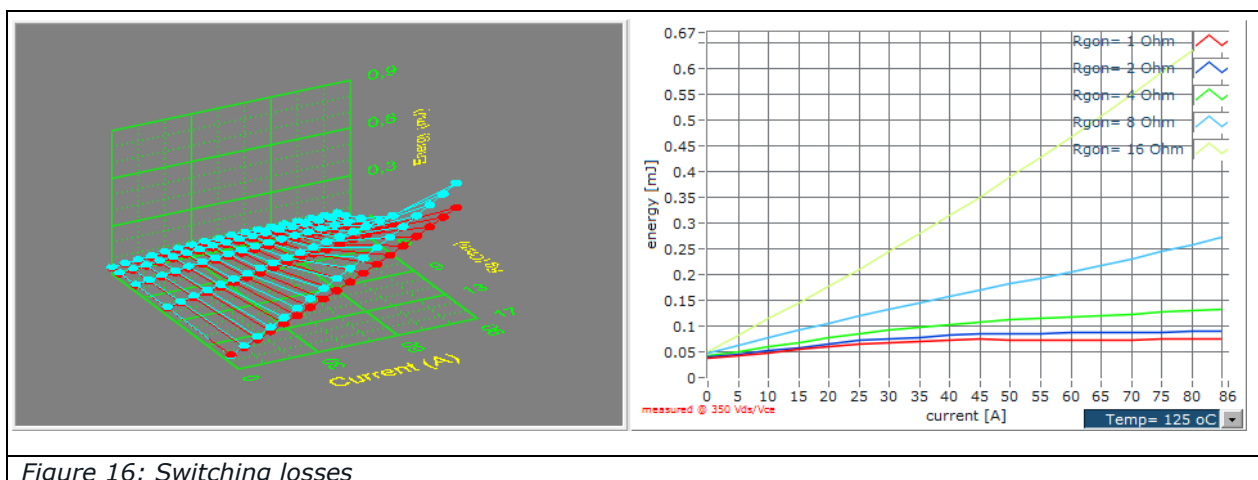


Each device's static loss is calculated on the basis of output characteristics $V = f(I)$ measured at 25 °C and 125 °C. The device's voltage drop at any other temperature is calculated using the linear interpolation method.



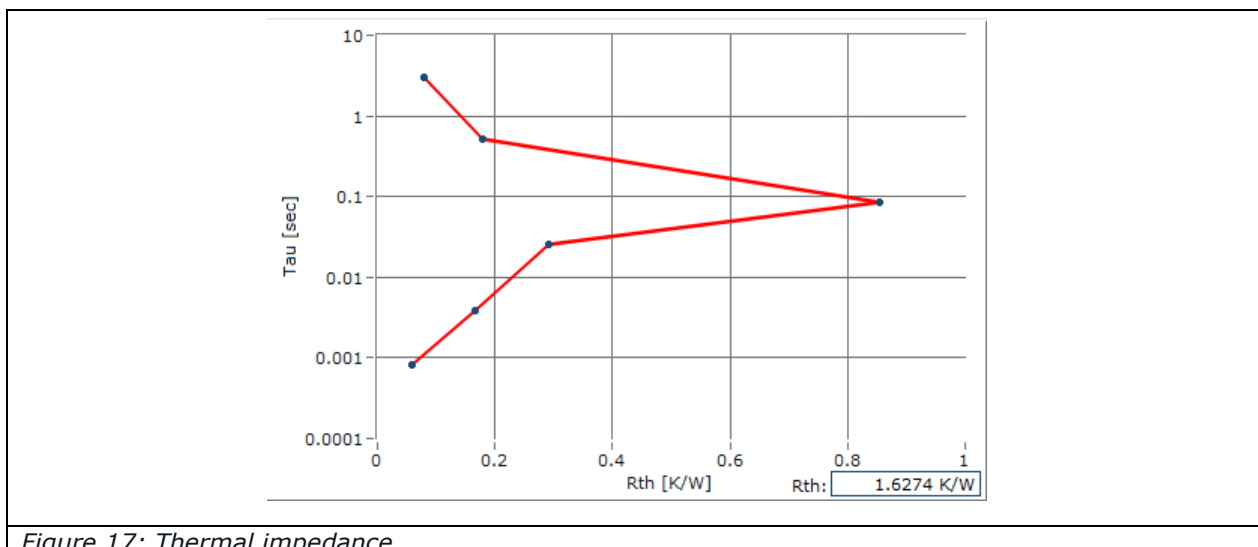
Switching losses are calculated by measuring E_{on} , E_{off} and E_{rec} . Measurements were taken at 25 °C and 125 °C with discrete values of I_c and R_g .

Left-click and hold the 3D graph to turn the model in any direction. Zoom in and out using the mouse wheel.



The heat-sink temperature has to be fixed first for thermal modeling. The default value is $T_{sink}=80$ °C. The calculated total power loss is applied to a serial R-C thermal model to determine chip temperature.

The graph represents Tau over R_{th} .



All axes can be adjusted by double clicking on a value and retyping it. A change of the highest axis value performs a zoom in or out, a change of another values changes the step range.

6.2 flowSIM

The *flowSIM* tool calculates Vincotech's power modules for industrial drive applications. Its user interface looks much like that of the *flowSOL* tool. The top panel consists of the module selector, function boxes, entry mask and simulation section; the bottom panel presents the output graphs.

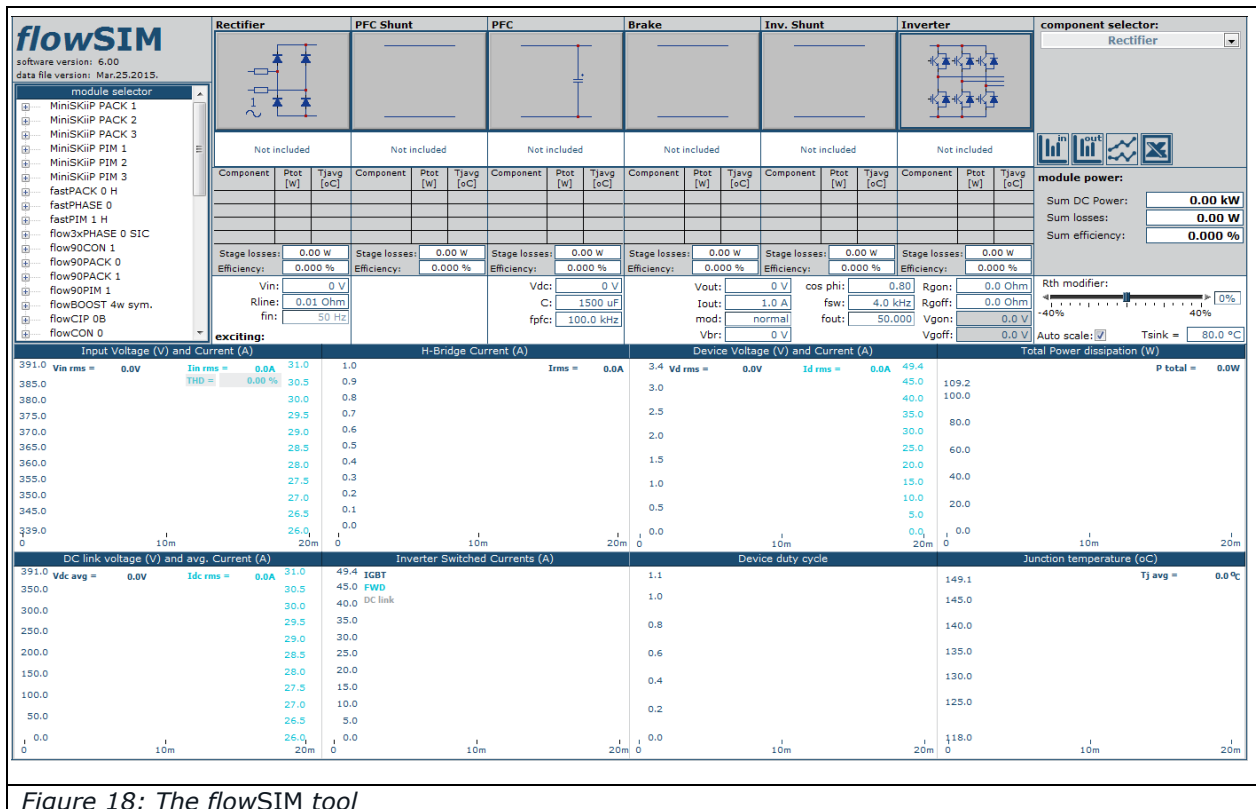


Figure 18: The *flowSIM* tool

The module selector lists all power modules designed for industrial drives. If you preselected modules earlier on the start screen, only those modules will be listed.

The *flowSIM* tool provides six function boxes labeled Rectifier, PFC Shunt, PFC, Brake, Inverter Shunt and Inverter. If the module is configured to perform one of these functions, the corresponding box will present the applicable circuit diagram framed in polar blue. If not, the box will be empty and framed in dark blue. The selected power module's full name is indicated below the enabled boxes.

A right-click on an enabled box opens the module selector. It lists all power modules that you can substitute to perform this function. In this case, all parameter changes are retained as described for the *flowSOL* tool.

Right-clicking a disabled function box also opens the module selector. In this case, a combination of different power modules, say a rectifier and an inverter, can be simulated.

Left-clicking any function box, enabled or disabled, changes the configuration of that part of the circuit.

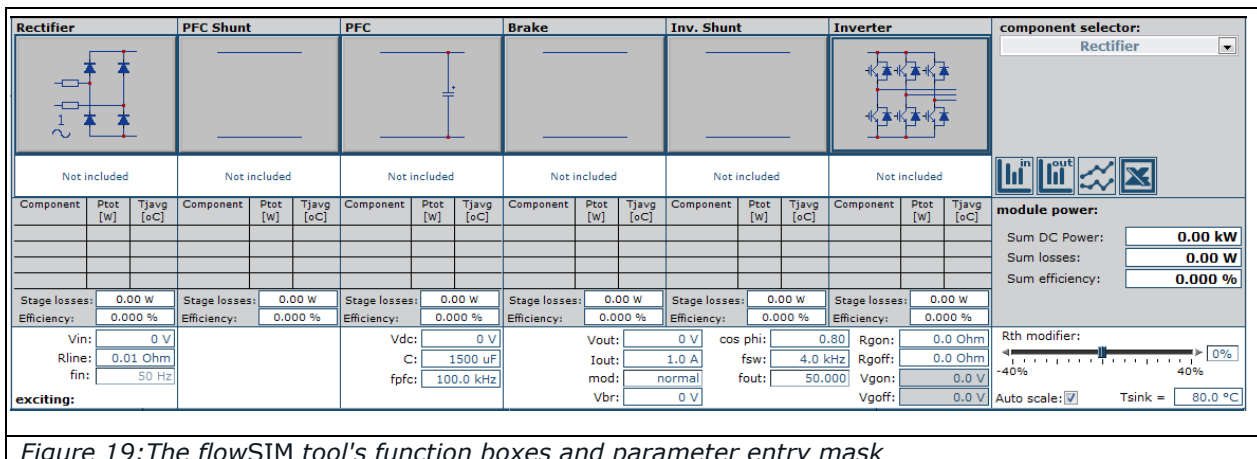
Here's an example of what you can do with a disabled block:

Say you want to simulate a six-pack inverter and need to define the type of input voltage to this end. You can choose DC voltage, one phase rectified or three phases rectified, and then adjust the voltage value, line resistance and – depending on your selection – the frequency.

Here's an example for an enabled block:

If you're simulating a six-pack, you could change the type of application from three-phase to one-phase.

Adjust the parameters after you have configured the module. Several are available depending on the enabled function box and selected power module.



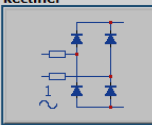
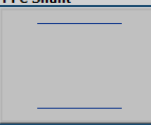
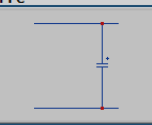
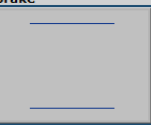
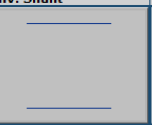
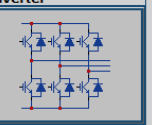
Rectifier	PFC Shunt	PFC	Brake	Inv. Shunt	Inverter	component selector:
						Rectifier
Not included	Not included	Not included	Not included	Not included	Not included	
Component	Ptot [W]	Tjavg [°C]	Component	Ptot [W]	Tjavg [°C]	Component
Stage losses: 0.00 W	Stage losses: 0.00 W	Stage losses: 0.00 W	Stage losses: 0.00 W	Stage losses: 0.00 W	Stage losses: 0.00 W	Stage losses: 0.00 W
Efficiency: 0.000 %	Efficiency: 0.000 %	Efficiency: 0.000 %	Efficiency: 0.000 %	Efficiency: 0.000 %	Efficiency: 0.000 %	Efficiency: 0.000 %
Vin: 0 V		Vdc: 0 V	Vout: 0 V	cos phi: 0.80	Rgon: 0.0 Ohm	Rth modifier: 0%
Rline: 0.01 Ohm		C: 1500 uF	Iout: 1.0 A	fsw: 4.0 kHz	Rgoff: 0.0 Ohm	
fin: 50 Hz		fpfc: 100.0 kHz	mod: normal	fout: 50.000	Vgon: 0.0 V	
exciting:			Vbr: 0 V		Vgoff: 0.0 V	Auto scale: <input checked="" type="checkbox"/> Tsink = 80.0 °C

Figure 19: The flowSIM tool's function boxes and parameter entry mask

- Rectifier

- V_{in} - input voltage in [V]

Input voltage is specified as a line-to-line RMS value for three-phase devices and as a line-to-ground RMS value for single-phase or DC applications. The default is DC if the module lacks a rectifier, but this can be changed to single-phase or three-phase.

- R_{line} - line resistance in [Ω]
- f_{in} - frequency of the input voltage in [Hz]

- PFC

- V_{dc} - output voltage of the PFC stage in [V]

For a PFC simulation, V_{dc} must be higher than V_{in_peak} :

$$V_{in_peak} = V_{in} \cdot \sqrt{2}$$

- C - DC link capacitance in [μF]

AC ripple on the DC link voltage is determined by DC link capacitance.

- f_{pfc} - switching frequency of the PFC stage in [Hz]

A virtual six-pack is added to the configuration if only a PFC stage is to be simulated. It serves as a load for the PFC even if the selected module does not include a six-pack. (The output parameters field is enabled even for a virtual six-pack.)

- Brake

- V_{br} - brake voltage in [V]

Brake voltage must be equal to or higher than the rectified input voltage V_{in} .

- Inverter

- V_{out} - output voltage in [V]

(first harmonic of the output voltage under nominal operating conditions)

Output voltage is specified as a line-to-line RMS value for three-phase devices and as a line-to-ground RMS value for single-phase or DC applications.

- I_{out} - RMS output current in [A]

The output current can be specified in positive values for motor applications and negative values for generator mode. In generator mode, calculations are made on the basis of a DC link voltage V_{dc} equal to the brake voltage. V_{br} must be equal to or higher than the rectified input voltage V_{in} .

- $\cos \phi$ - efficiency factor $\cos \phi$, range 0 to 1

- f_{sw} - switching frequency in [Hz]

- mod - modulation index

the 3rd harmonic can be added to the modulator signal

- f_{out} – first harmonic output voltage frequency in [Hz]
only preset values can be selected
 - R_{gon} – switch-on gate resistor in [Ω]
 - R_{goff} – switch-off gate resistor in [Ω]
The value must be equal to or lower than switch on-gate resistor R_{gon} .
 - V_{gon} – positive gate driver voltage in [V] (This preset is nonadjustable)
 - V_{goff} – negative gate driver voltage in [V] (This preset is nonadjustable)
- T_{sink} – heat sink temperature in [$^{\circ}C$]

DC power is indicated below the simulation section and calculated as follows:

$$P_{DC} = \sqrt{3} \cdot V_{line} \cdot I_{out} \cdot \cos\varphi \quad \text{Three-phase applications}$$

$$P_{DC} = V_{line} \cdot I_{out} \cdot \cos\varphi \quad \text{One-phase applications}$$

The graphs in the lower *flowSIM* panel are updated after every parameter change. These graphs are windows into the simulation affording you views of input voltage curves, input current, DC link values and the like to show how individual components respond to the given parameters. Select the component you wish to analyze in the component selector box at the right of the top panel.

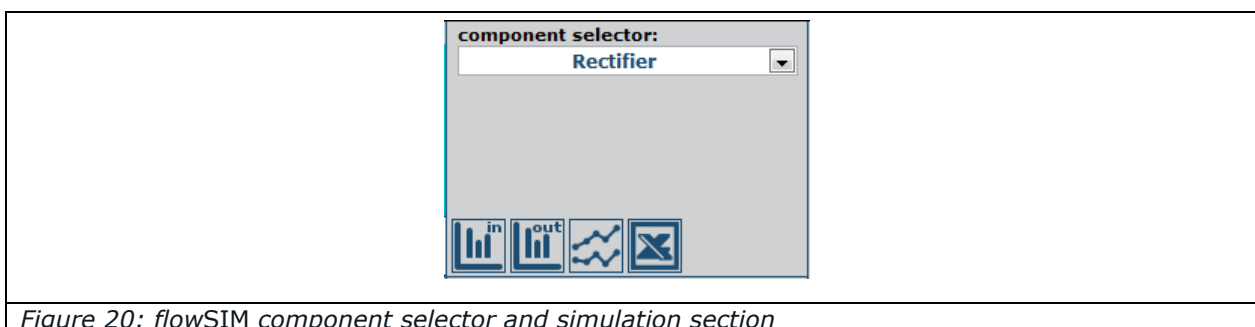


Figure 20: *flowSIM* component selector and simulation section

Click these three buttons to get full views of the selected component's input and output waveforms, losses and switching characteristics, respectively.



Detailed input and output voltage and current waveforms
(independent of the selected component)

Input and
rectifier

DC link and PFC

Output

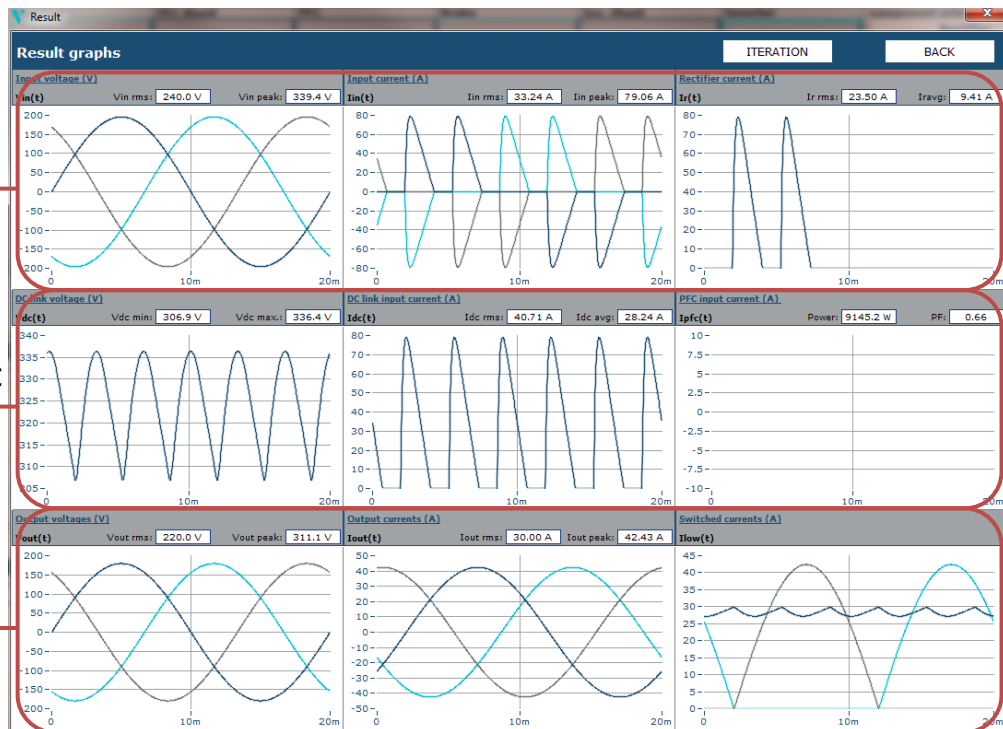


Figure 21: The flowSIM tool's detailed input and output waveforms



Detailed static and dynamic losses and temperature graphs of the selected component

Current, voltage



Duty cycle, temperature rise

Power losses, junction temperature, thermal resistance

Power losses, junction temperature

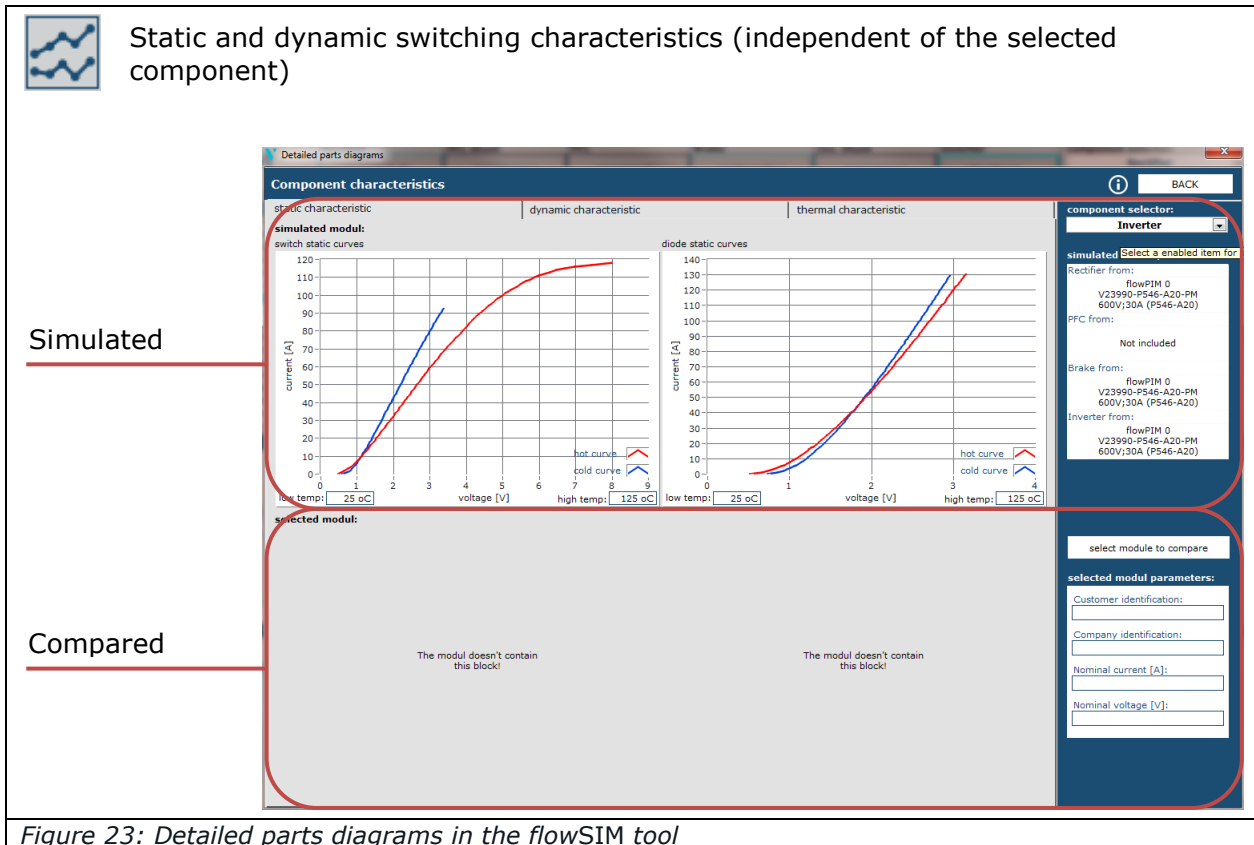
Figure 22: Detailed static and dynamic losses and temperature graphs in the flowSIM tool

The 'Detailed parts diagrams' window serves to analyze static and dynamic switching characteristics and thermal impedance. Informative graphs will be shown on various pages depending on the selected module's configuration as a rectifier, PFC or inverter.

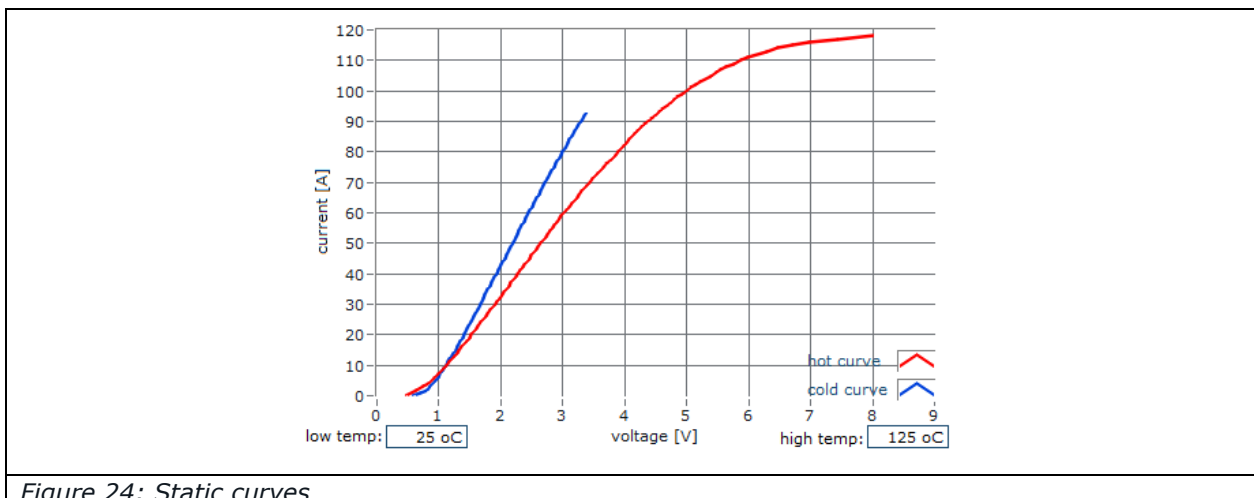
You can also select a second power module to compare two options directly. To do this, open the module selector by clicking the 'select module to compare' button on the left. A second module is then simulated using the same set of parameters as for the first.



Static and dynamic switching characteristics (independent of the selected component)

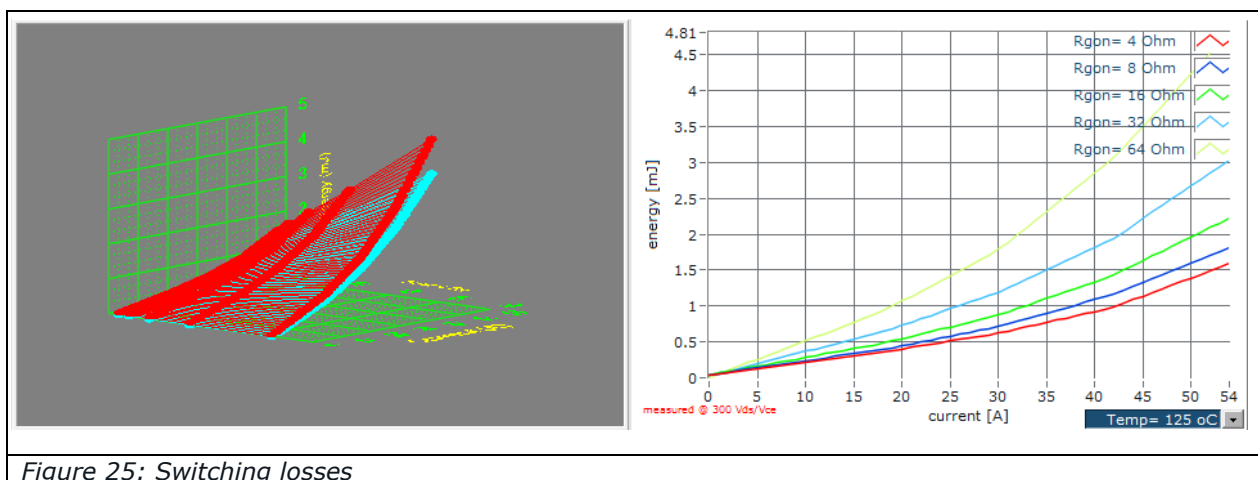


Each device's static loss is calculated on the basis of output characteristics $U = f(I)$ measured at 25 °C and 125 °C. The device's voltage drop at any other temperature is calculated using the linear interpolation method.

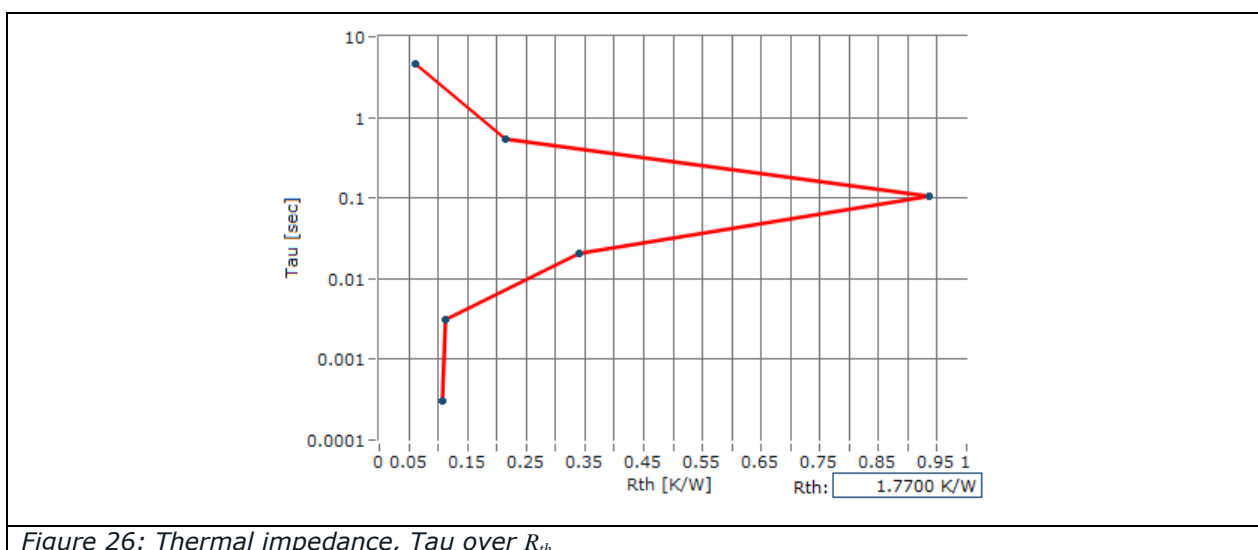


Switching losses are calculated by measuring E_{on} , E_{off} and E_{rec} . Measurements were taken at 25 °C and 125 °C with discrete values of I_c and R_g .

Left-click and hold the 3D graph to turn the model in any direction. Zoom in and out using the mouse wheel.



The heat-sink temperature has to be fixed first for thermal modeling. The default value is $T_{sink}=80$ °C. The calculated total power loss is applied to a serial R-C thermal model to determine chip temperature.



All axes can be adjusted by double clicking on a value and retyping it. A change of the highest axis value performs a zoom in or out, a change of another values changes the step range.

6.3 flowSEL

If you need to find the right power module for an industrial drive application, *flowSEL* is the tool to use. All graphs are based on actual measurements, which is why *flowSEL* provides accurate data on current, power and temperature conditions that will help you identify the best solution.

The *flowSEL* screen is divided into three sections, the entry mask on the top left, the module repertory below it, and the graphs on the right.

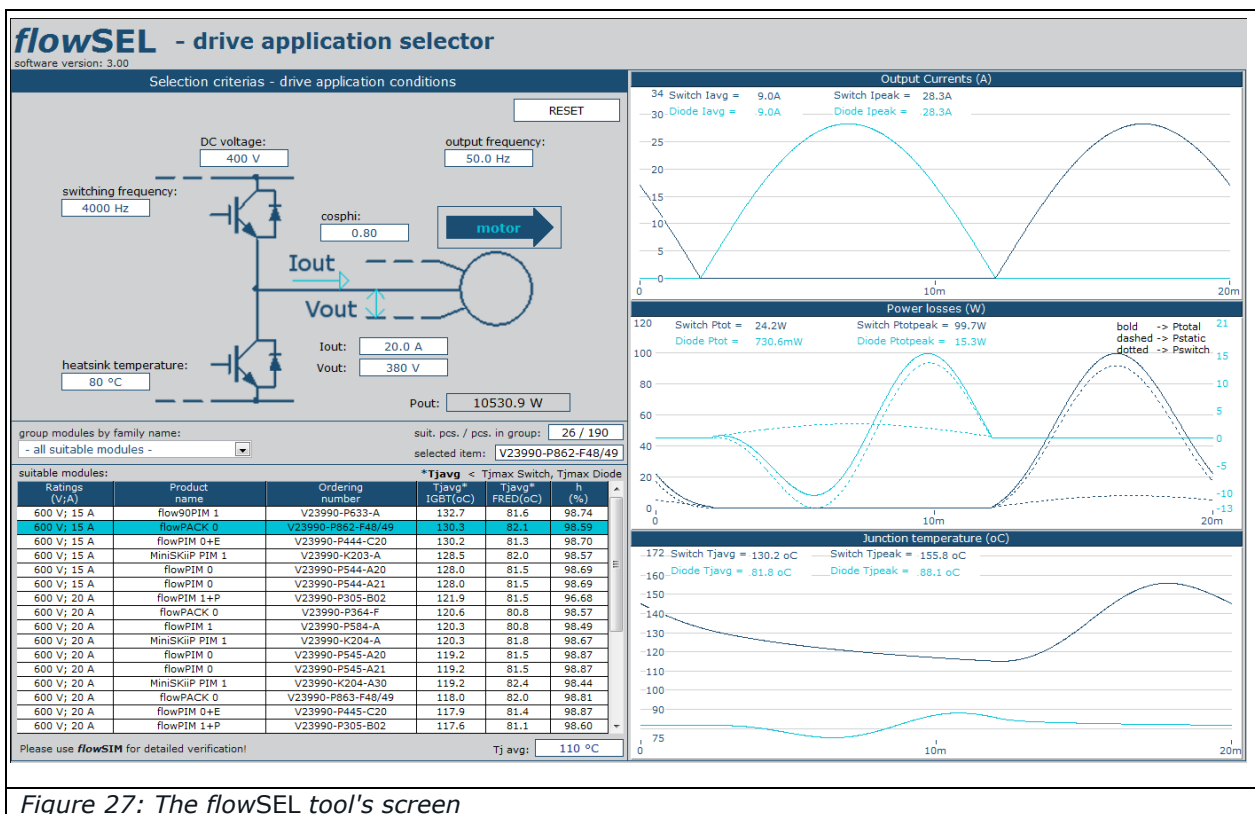


Figure 27: The *flowSEL* tool's screen

First open the drop-down menu under "Group modules by customer name" to select a module group and narrow down the list of available power modules. You can reduce this list further by choosing among the available housings and topologies. Then adjust the set of parameters to suit the application you have in mind. The list of suitable modules is refreshed automatically with every parameter change.



The adjustable parameters are:

- DC voltage in [V]
- Switching frequency in [Hz]
- Heat sink temperature in [°C]
- Output/grid frequency in [Hz]
- Efficiency factor $\cos \varphi$
- RMS output current in [A]
- RMS output voltage in [V]
- The switch on the right of the application diagram toggles between motor and generator operating modes.

The program calculates the average junction temperature of the switching element, the freewheeling diodes and the coefficient of efficiency. These values are displayed in the module table's right columns.

The lower limit for the average junction temperature may also be adjusted. Modules with less than $T_{j \text{ avg min}}$ are not displayed in the table. The default value is 110 °C.

The simulation starts automatically once a power module is selected.