Elite Pairing Studio

User Guide

Ver. 1.0

Outline of User Guide

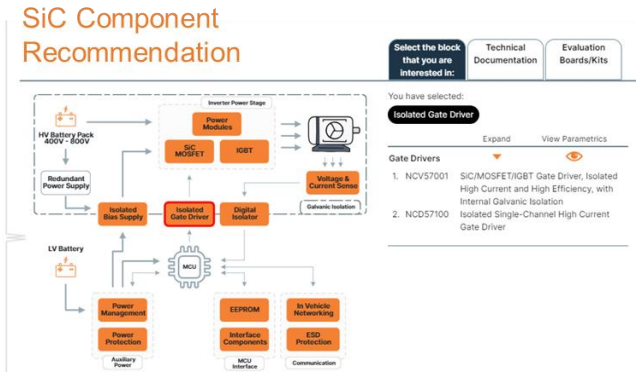
1	Introduction to Elite Pairing Studio
2	Stage 1: Choose
3	Stage 2: Simulate
4	Stage 3: Analyze

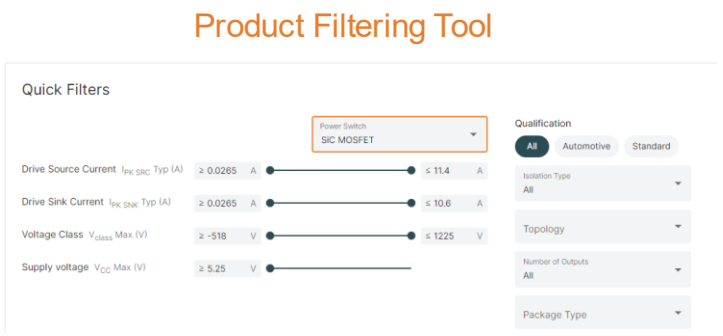
Outline of User Guide

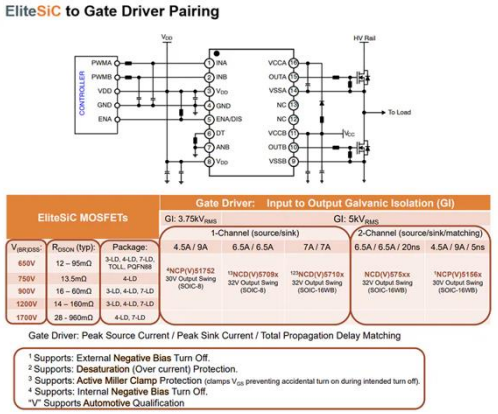
1	Introduction to Elite Pairing Studio
2	Stage 1: Choose
3	Stage 2: Simulate
4	Stage 3: Analyze

Introduction

- onsemi provides online services to assist in selecting the right Switch and Gate Driver (GD)
  - “Product Recommendation Tools+” that apply parametric filters to the existing product tables (see picture below)
  - Tables categorizing GD per switch voltage rate, isolation, package, strength, and number of channels (see picture below)
- While these tools offer a helpful initial indication of which GD may be suitable, they do not yet provide specific “Switch-GD” pairings, nor do they offer guidance on optimizing these pairings
- Circuit/application designer will have to perform a series of calculations/simulations to identify the most suitable driver and to implement it in the system. The main idea is that we will assist designers on that process.

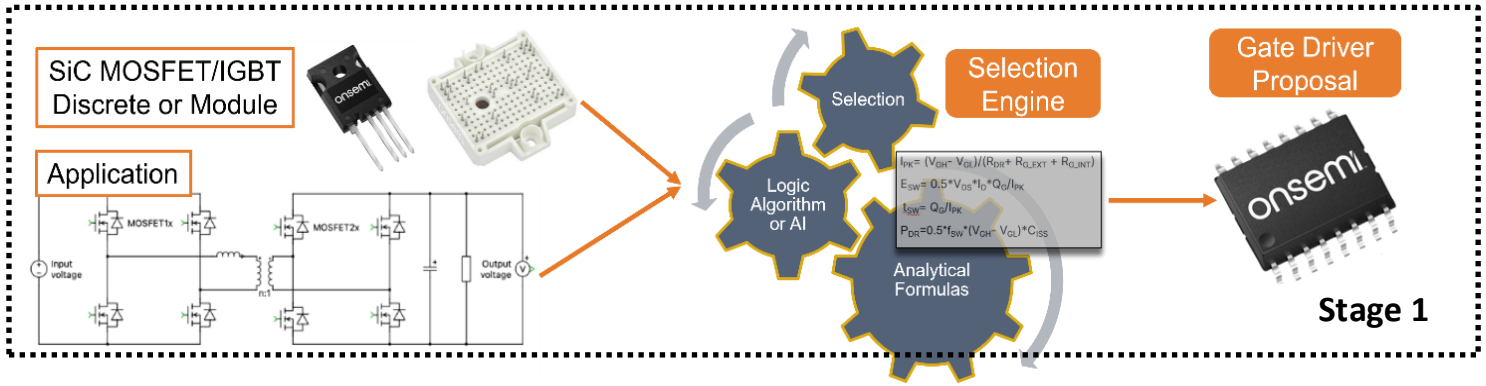






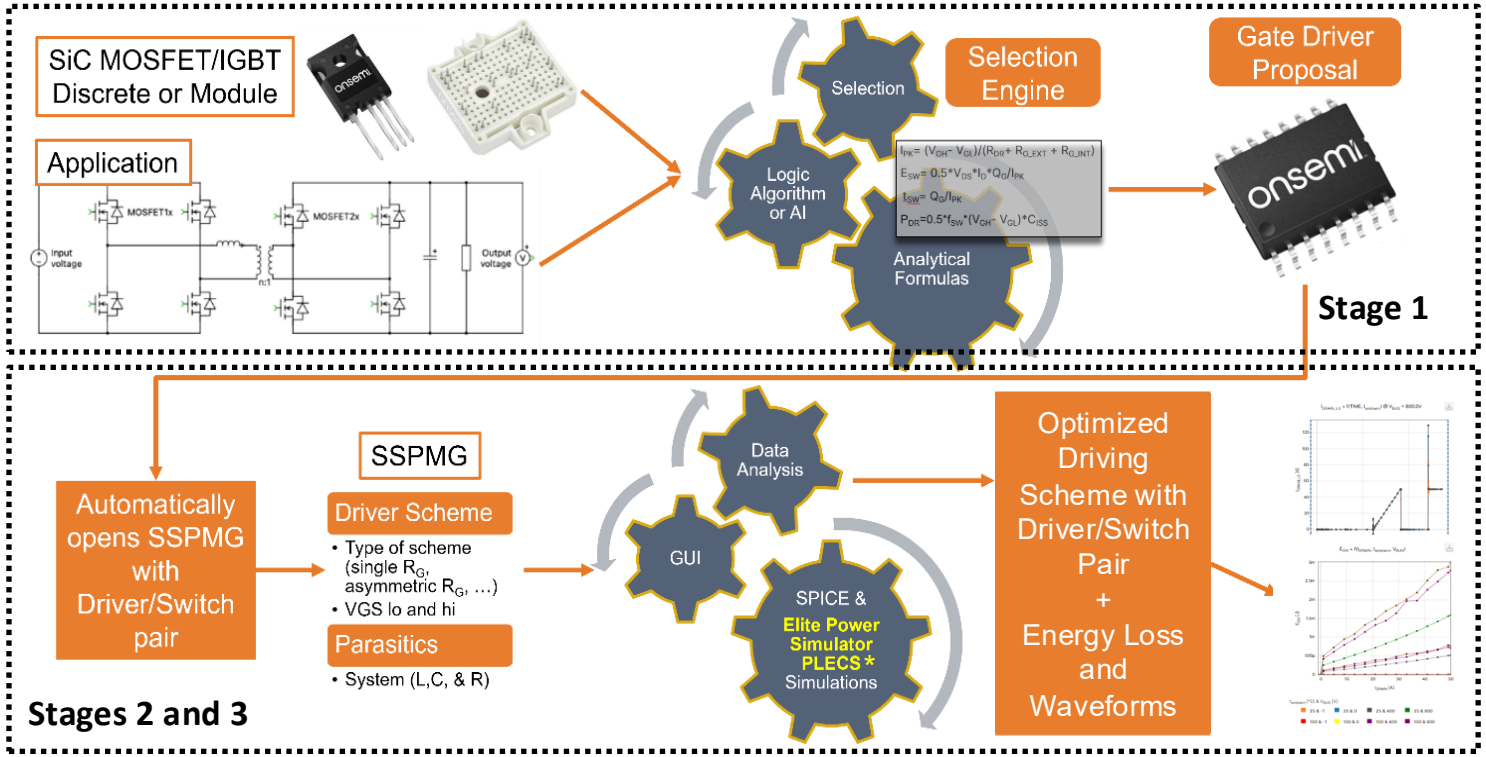
Introduction

- Elite Pairing Studio is a method to pair “power switches” with “gate drivers” and to optimize the “driving scheme” based on three sequential Stages
  - **Stage 1 (“Choose”)** – a group of GDs from a given portfolio are recommended after the execution of a series of algorithms and analytical formulas that receive user inputs and data from a parametric table for GDs and from a parametric table for “power switches”. A detailed description of the analytical models implemented at Stage 1 is found in the Application Note AND90421/D “Inside Elite Pairing Studio”.



Introduction

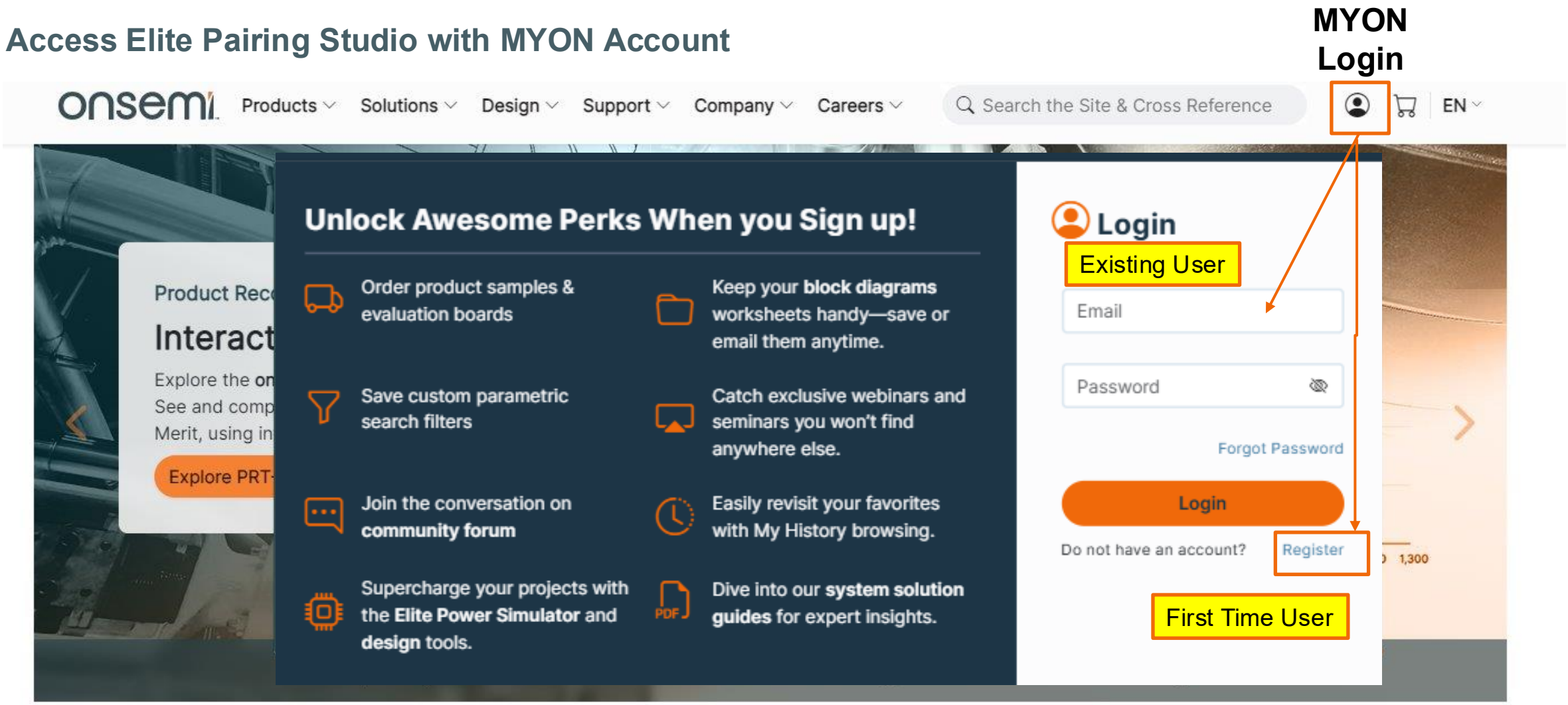
- Elite Pairing Studio is a method to pair “power switches” with “gate drivers” and to optimize the “driving scheme” based on three sequential Stages
  - **Stages 2 and 3 (“Simulate & “Analyze”)** – the SPICE models for selected “Switch-GD” pair are automatically inserted in a SPICE circuit scheme (Double Pulse Test or other) that includes the “driving scheme”. The user defines a parametric range (R_G, ambient temperature, I_D, V_{DC} …) to run dynamic simulations in the background. An analysis tool suite is finally used to manage and investigate the simulation results.



* Feature available in SSPMG

Introduction

Access Elite Pairing Studio with MYON Account



Outline of User Guide

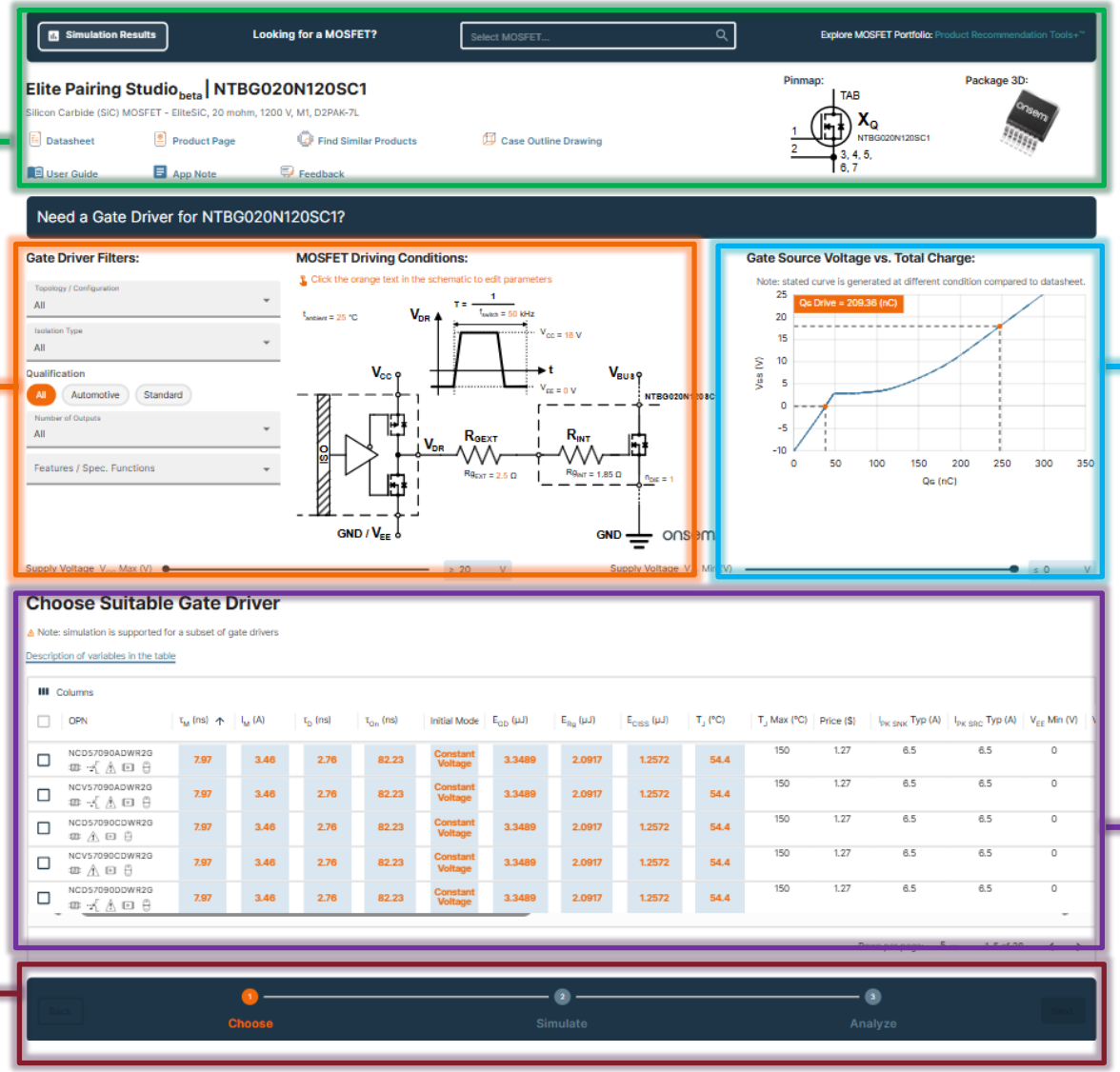
General View

On onsemi web page, in the main menu on page top select “Design”, and then, in the last column on the right, select “Elite Pairing Studio” or go directly to “www.onsemi.com/elite-pairing-studio”

Selected SiC MOSFET and corresponding information

User inputs for application, operating conditions, specific features and functionalities

Flowline between Stages



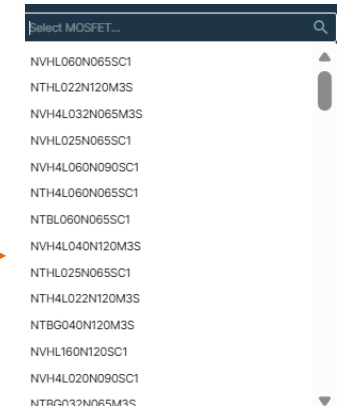
Gate charge curve for selected SiC MOSFET (extracted from SPICE model)

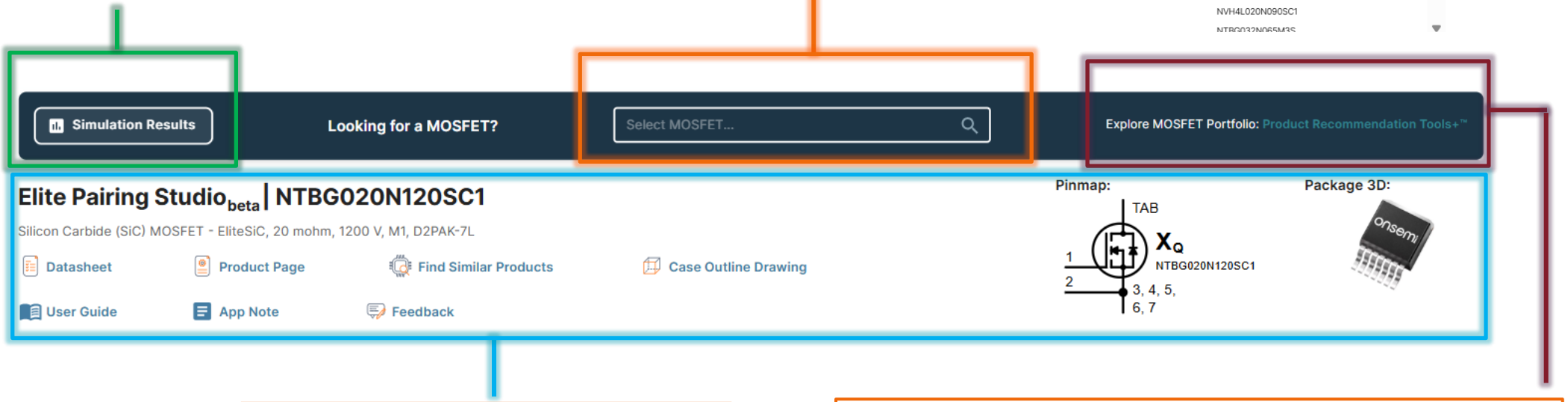
Output table with selectable GD list and prioritizing fields (price, T_{J_MAX}, I_{PK}, …)

Step 1: Choosing your SiC MOSFET

Only use this button to retrieve previous simulation results

Click this box to unfold onsemi's SiC MOSFET portfolio and select one in the list





Browse the information of the selected SiC MOSFET

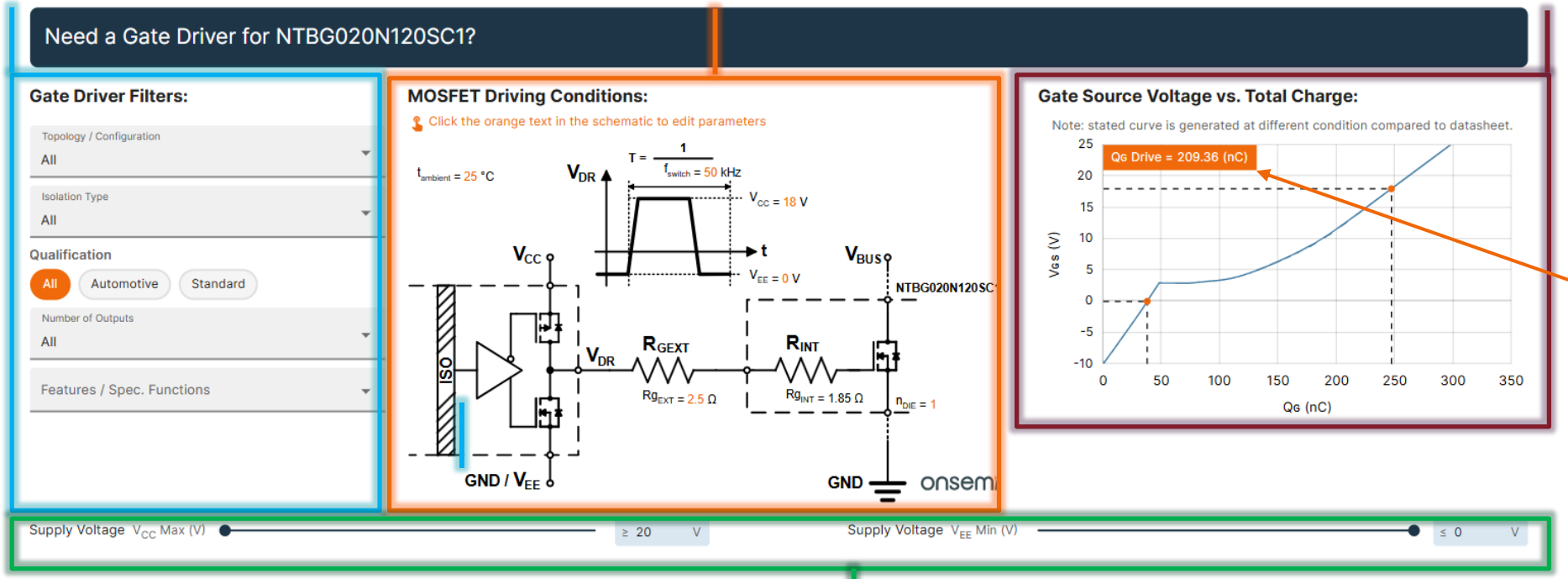
Link to "Product Recommendation Tools+" to help you choosing the right SiC MOSFET

Step 2: Providing information for SiC MOSFET gate driving

Select the features and functionalities that are needed for the GD

Click the orange text to change the default values for the GD operation

Gate charge curve for selected SiC MOSFET. Refreshes when selecting new SiC MOSFET and max/min driving voltage

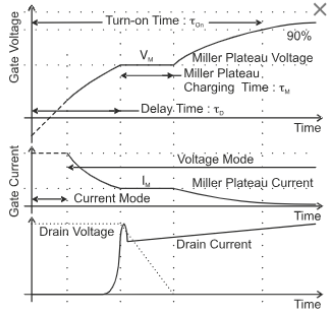


Q_G for the selected SiC MOSFET and GD operation

Set max/min values for V_{CC}/V_{EE}

Step 3: Displaying and prioritizing the fields of your selection

Click this text to quickly see a description of τ_M, I_M, τ_D and τ_{On}



Select ascendent or descendent order in your field of interest

Choose Suitable Gate Driver

Note: simulation is supported for a subset of gate drivers

Description of variables in the table

Columns

OPN	τ_M (ns)	I_M (A)	τ_D (ns)	τ_{On} (ns)	Initial Mode	E_{GD} (µJ)	E_{Rg} (µJ)	E_{CISS} (µJ)	T_J (°C)	T_J Max (°C)	Price ($)	$I_{PK\ SNK}$ Typ (A)	$I_{PK\ SRC}$ Typ (A)	V_{EE} Min (V)
NCD57090ADWR2G	7.97	3.46	2.76	82.23	Constant Voltage	3.3489	2.0917	1.2572	54.4	150	1.27	6.5	6.5	0
NCV57090ADWR2G	7.97	3.46	2.76	82.23	Constant Voltage	3.3489	2.0917	1.2572	54.4	150	1.27	6.5	6.5	0
NCD57090CDWR2G	7.97	3.46	2.76	82.23	Constant Voltage	3.3489	2.0917	1.2572	54.4	150	1.27	6.5	6.5	0
NCV57090CDWR2G	7.97	3.46	2.76	82.23	Constant Voltage	3.3489	2.0917	1.2572	54.4	150	1.27	6.5	6.5	0
NCD57090DDWR2G	7.97	3.46	2.76	82.23	Constant Voltage	3.3489	2.0917	1.2572	54.4	150	1.27	6.5	6.5	0

Rows per page: 5 1–5 of 38

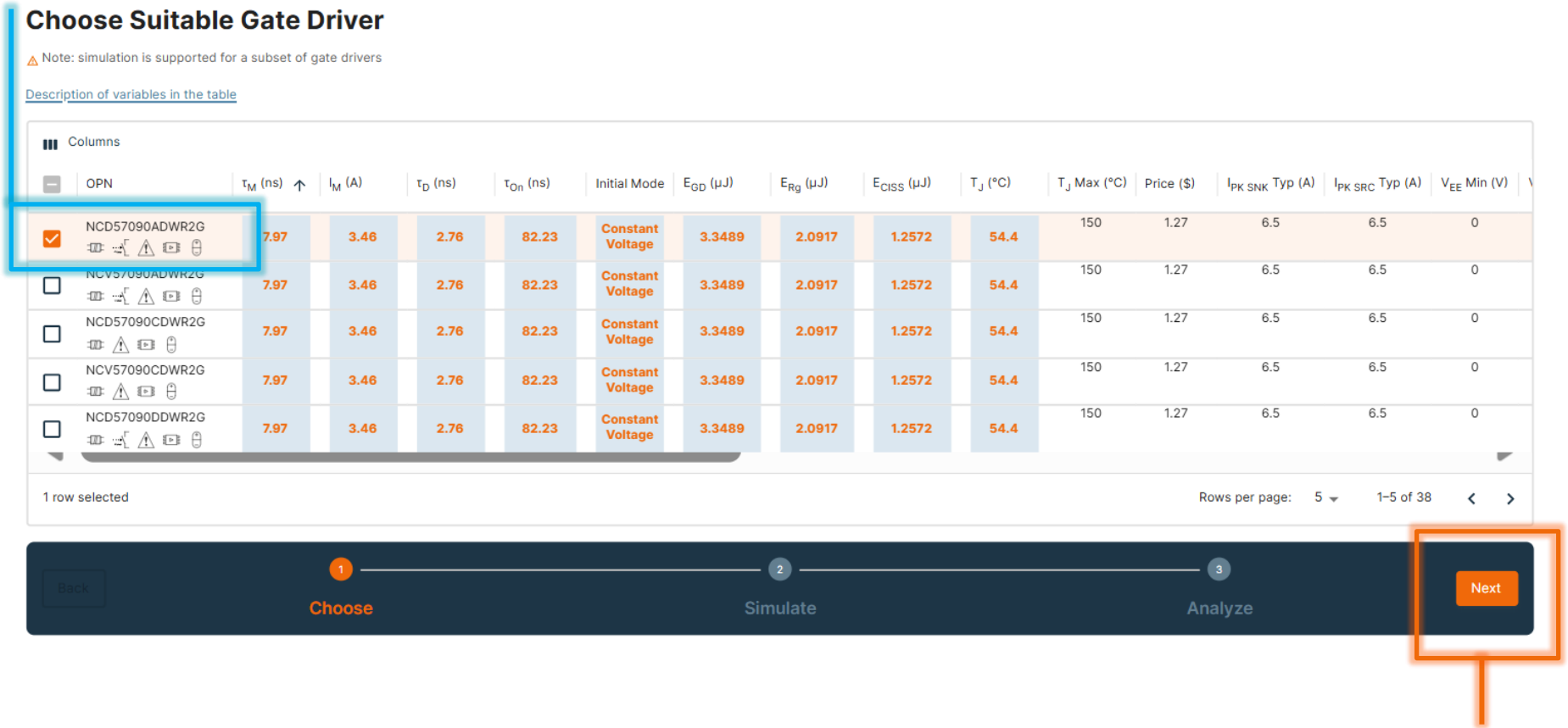
Back 1 Choose 2 Simulate 3 Analyze Next

Grey columns indicate values extracted from analytical formulas (see App. Note AND90421/D)

Table refreshes by modifying the inputs in Step 2

Step 4: Selecting your Gate Driver and moving to “Simulate”

Make your GD selection by clicking on the checkbox. Note that only the GDs with an available SPICE model are selectable (black ink).



Click Next in the flowline bar

Outline of User Guide

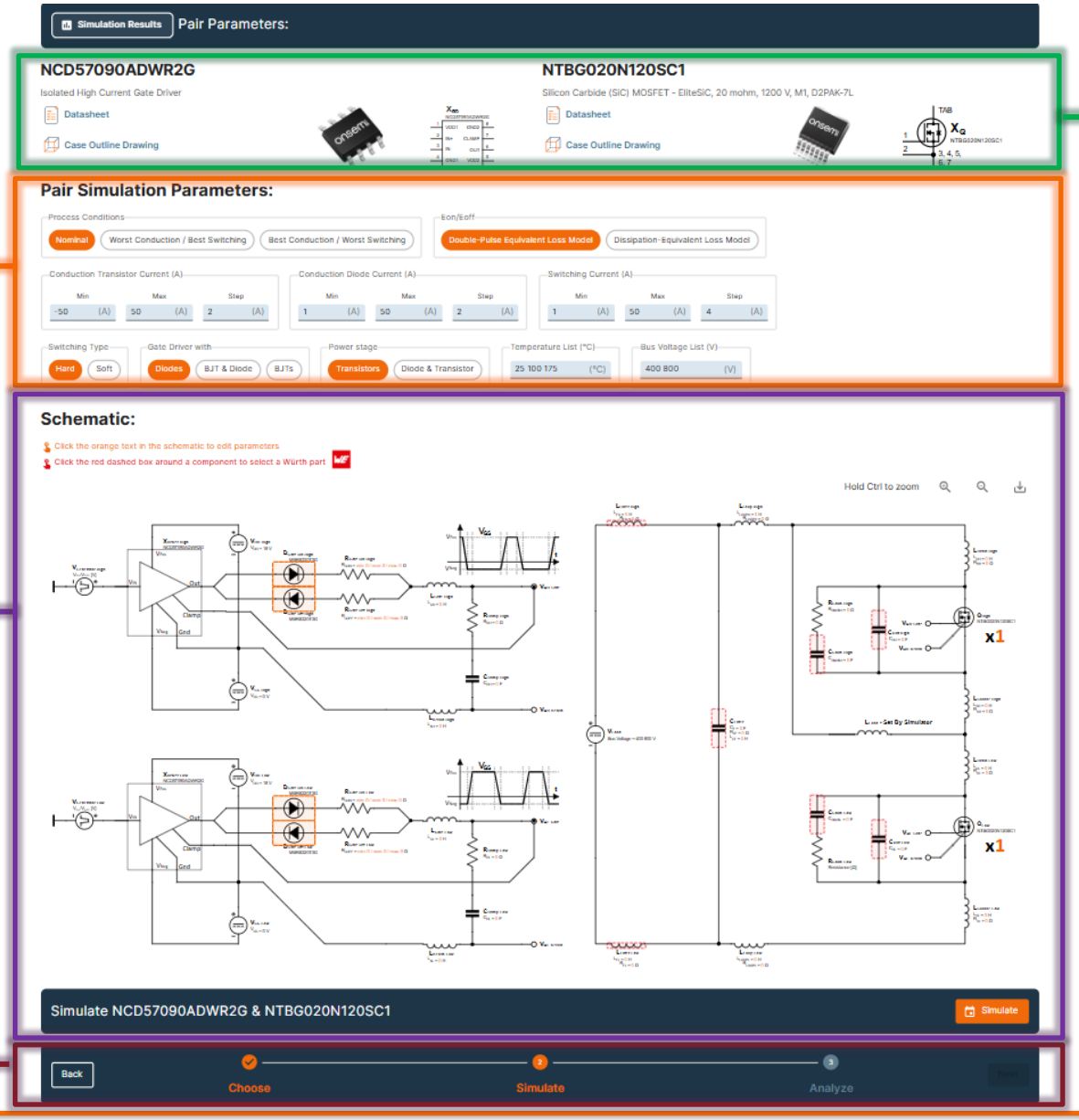
1	Introduction to Elite Pairing Studio
2	Stage 1: Choose
3	Stage 2: Simulate
4	Stage 3: Analyze

General View

Selector of circuit and parametric range for the SPICE simulations

Interactive circuit scheme to define circuit components and parasitic elements

Flowline between Stages

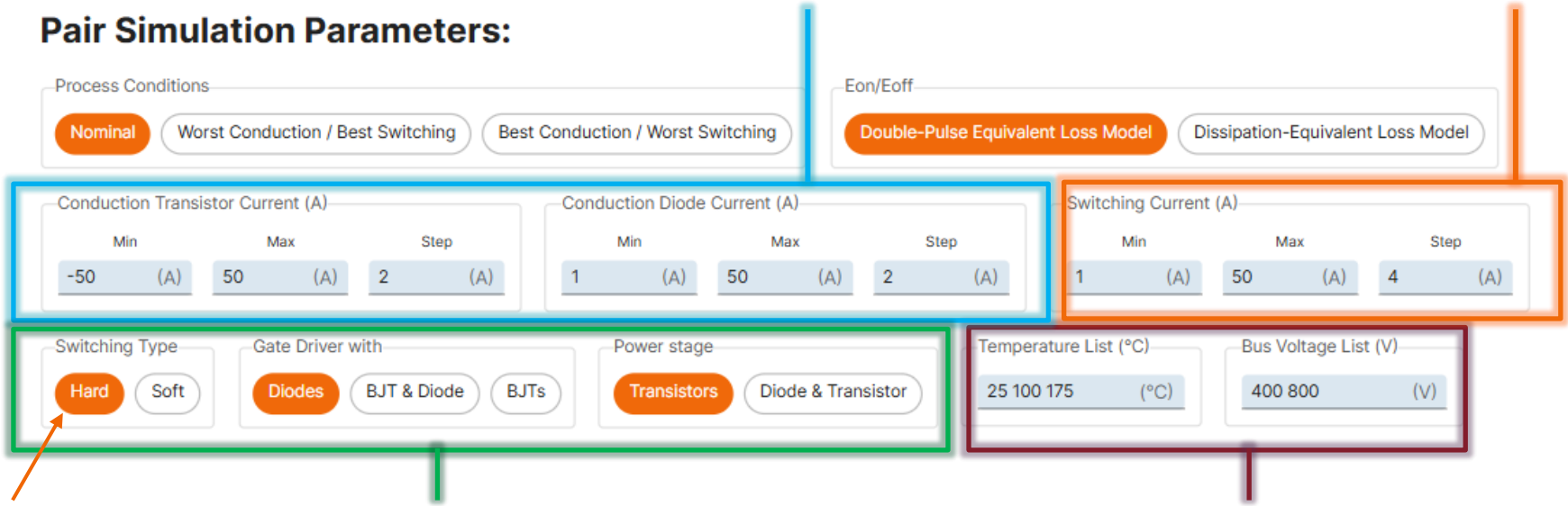


Selected “Switch-GD” pair and corresponding information

Step 1: Defining circuit and parameters

Fill in min/max currents and step to generate IdVds curves in DC mode

Fill in min/max currents and step to run dynamic SPICE simulations



"Hard" option for the conventional Double Pulse Test (DPT) scheme

Select the type of circuit. When modified, updates circuit scheme view

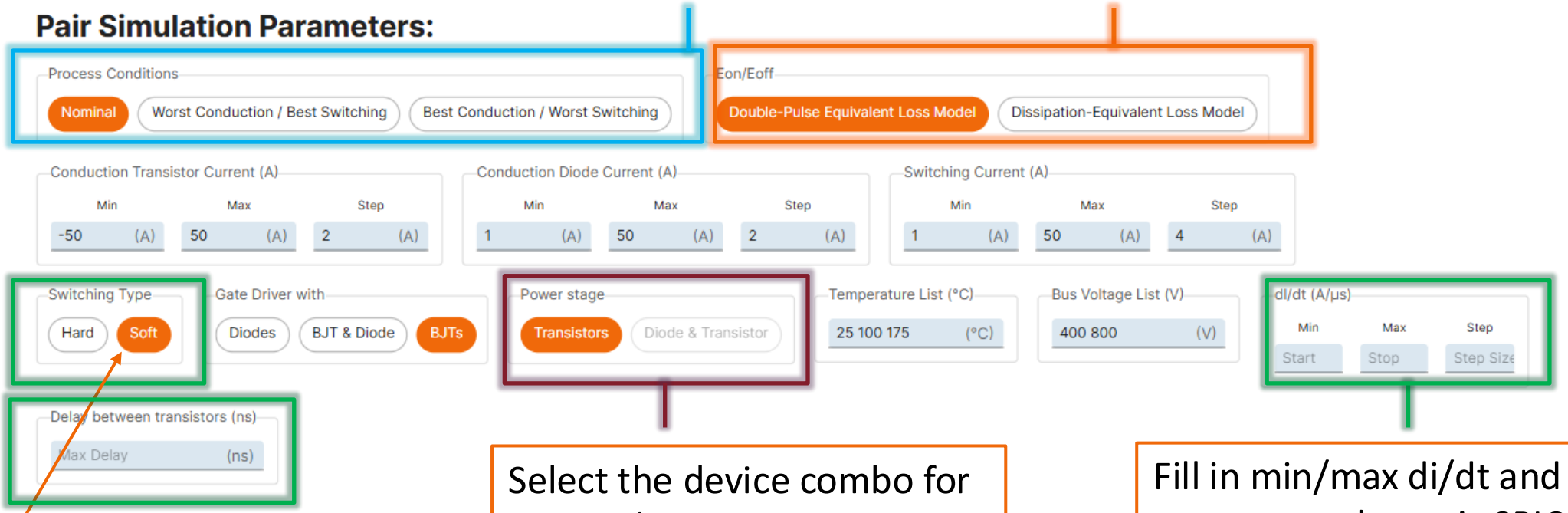
Fill in ambient temperature and bus voltage values to run dynamic SPICE simulations

Step 1: Defining circuit and parameters (cont.)

*More information on the methodology to simulate and extract switching losses is available in SSPMG User Guide ver.10

Select corner models in case they are available

Select the method to extract switching losses*

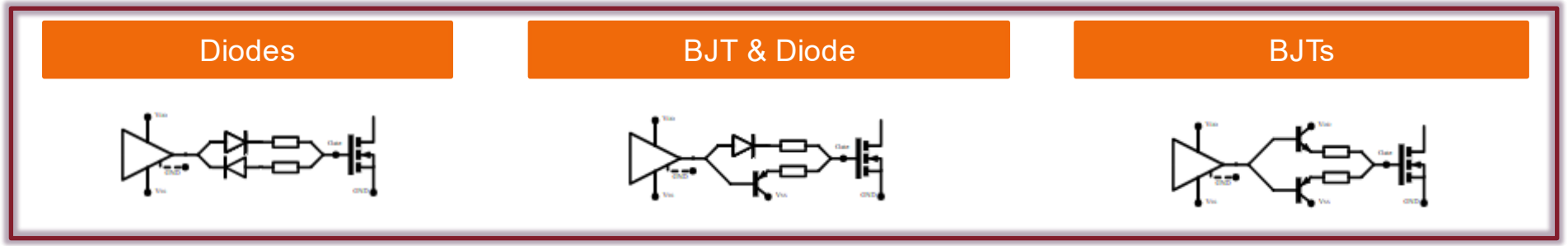


"Soft" option adds filter capacitors to the DPT scheme*

Select the device combo for control/synchronous positions in the power stage

Fill in min/max di/dt and step to run dynamic SPICE simulations ("Soft" option)

Step 1: Defining circuit and parameters (cont.)



Pair Simulation Parameters:

Process Conditions: Nominal | Worst Conduction / Best Switching | Best Conduction / Worst Switching

Eon/Eoff: Double-Pulse Equivalent Loss Model | Dissipation-Equivalent Loss Model

	Min	Max	Step
Conduction Transistor Current (A)	-50 (A)	50 (A)	2 (A)
Conduction Diode Current (A)	1 (A)	50 (A)	2 (A)
Switching Current (A)	1 (A)	50 (A)	4 (A)

Switching Type: Hard | Soft

Gate Driver with: Diodes | BJT & Diode | BJTs

Power stage: Transistors | Diode & Transistor

Temperature List (°C): 25 100 175 (°C)

Bus Voltage List (V): 400 800 (V)

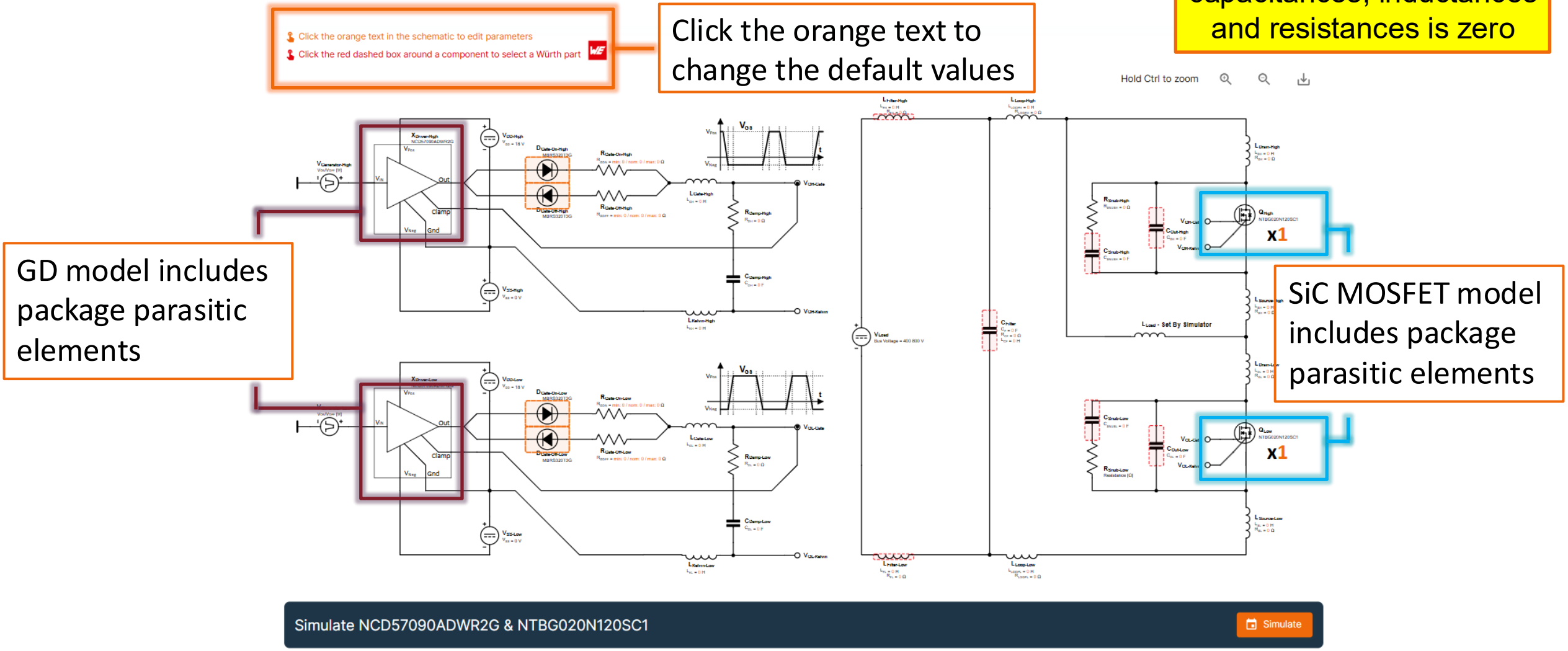
dI/dt (A/µs): Min: Start | Max: Stop | Step: Step Size

Delay between transistors (ns): Max Delay (ns)

Selects the type of device for asymmetric driving

Step 2: Setting components and parasitic elements

Default value for capacitances, inductances and resistances is zero

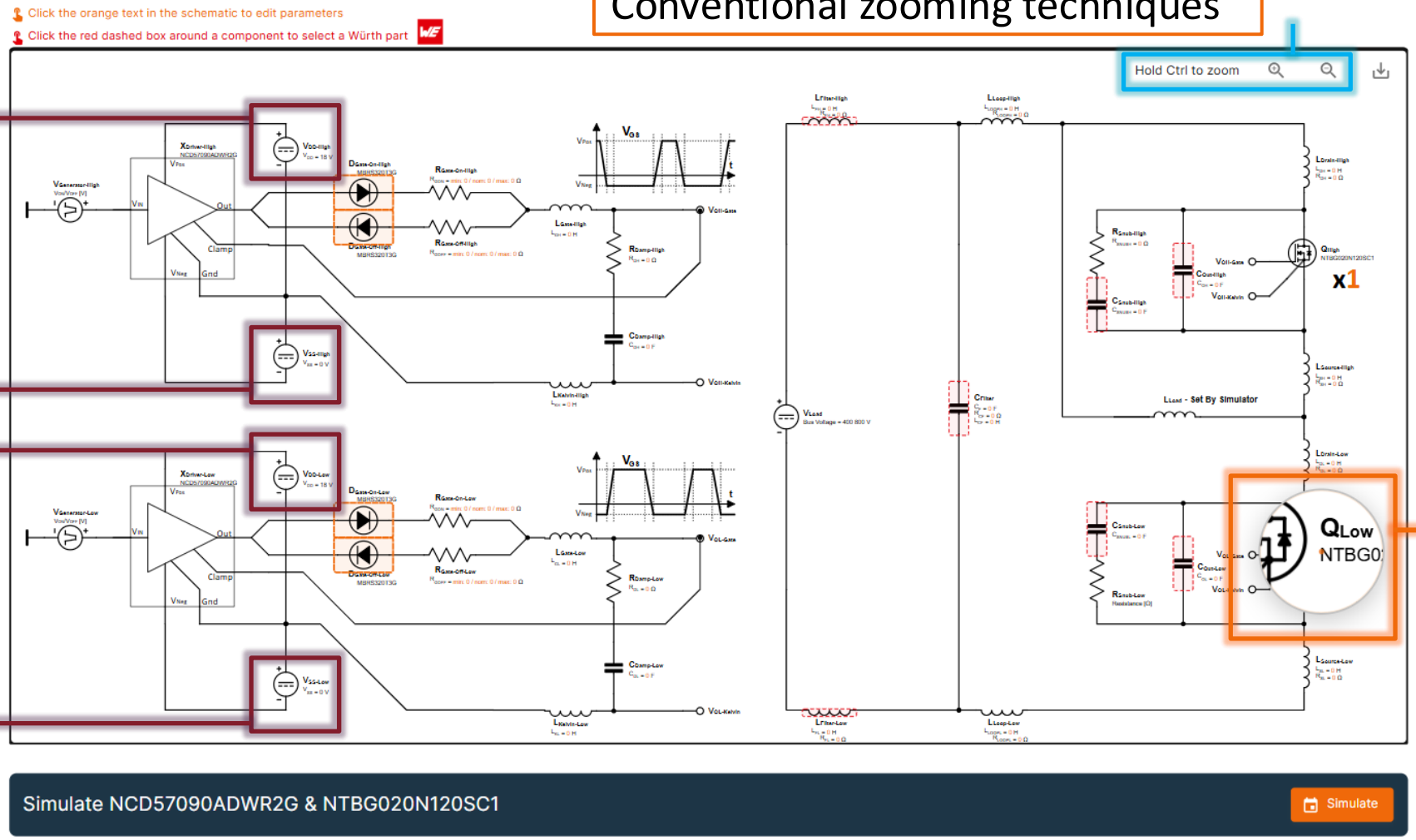


Step 2: Setting components and parasitic elements (cont.)

Conventional zooming techniques

Parameters inherited from Stage 1

Parameters inherited from Stage 1

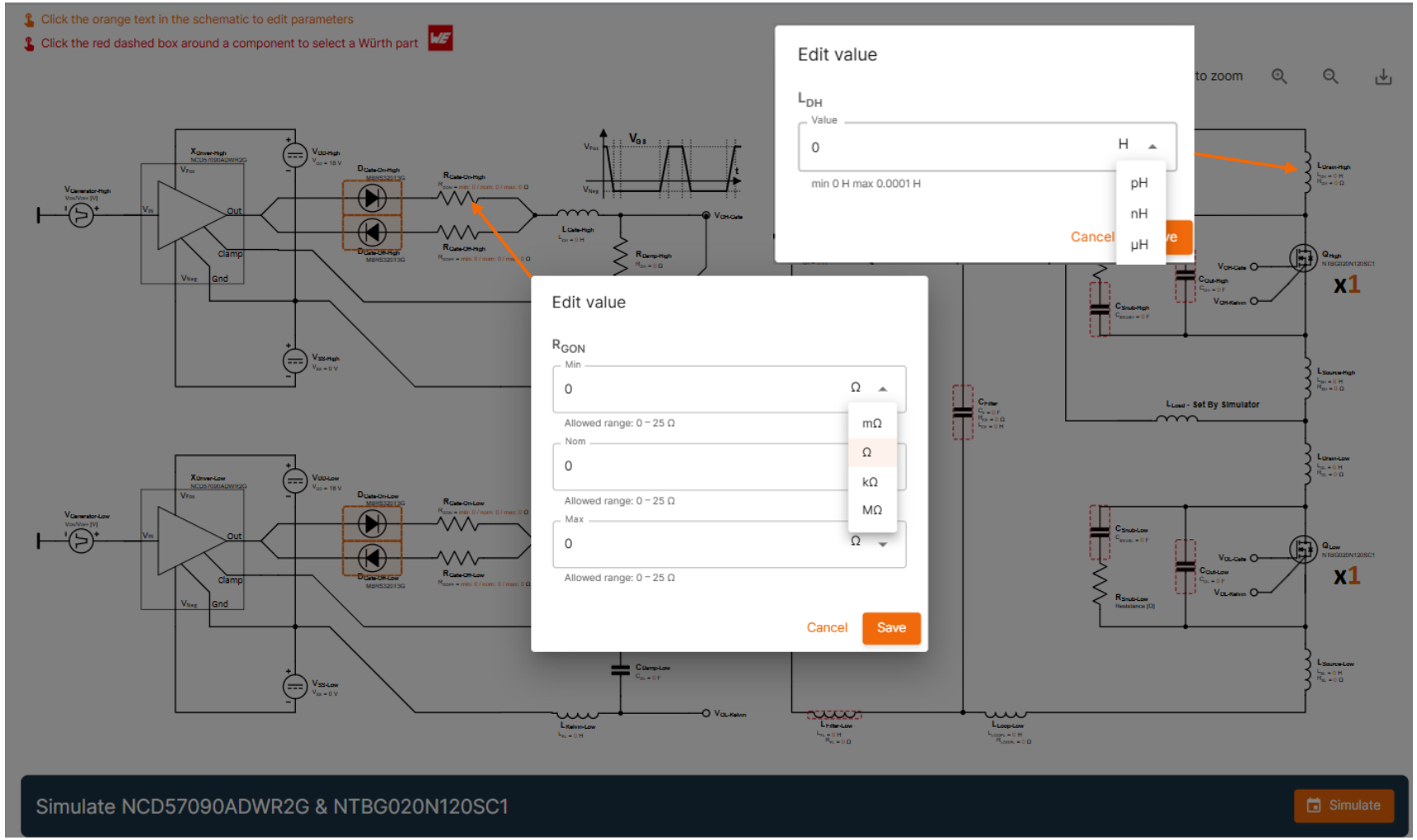


Magnifying lens is active by default

Step 2: Setting components and parasitic elements (cont.)

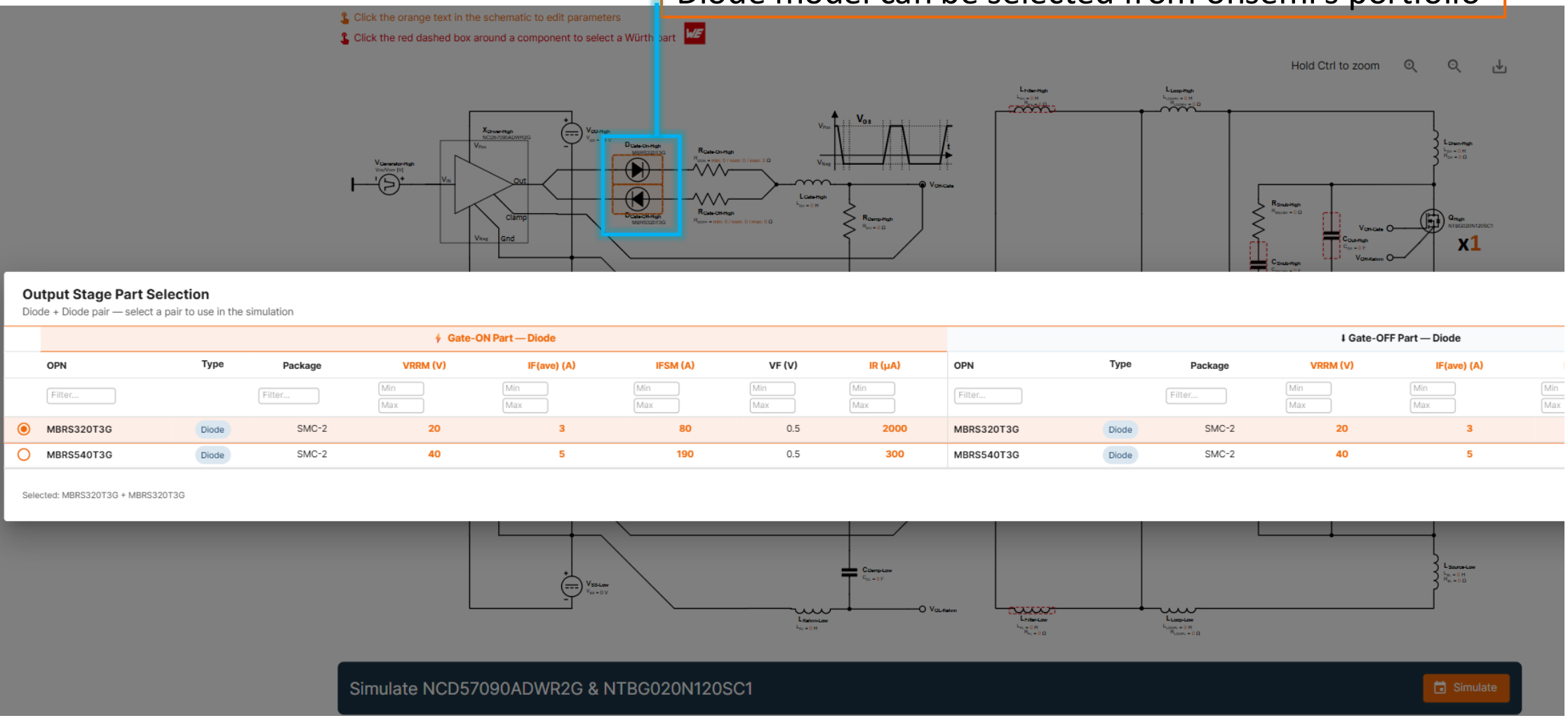
After clicking the orange text, an edit window opens

The edit window allows to select a single value for parasitic elements or a range for on/off gate resistances



Step 2: Setting components and parasitic elements (cont.)

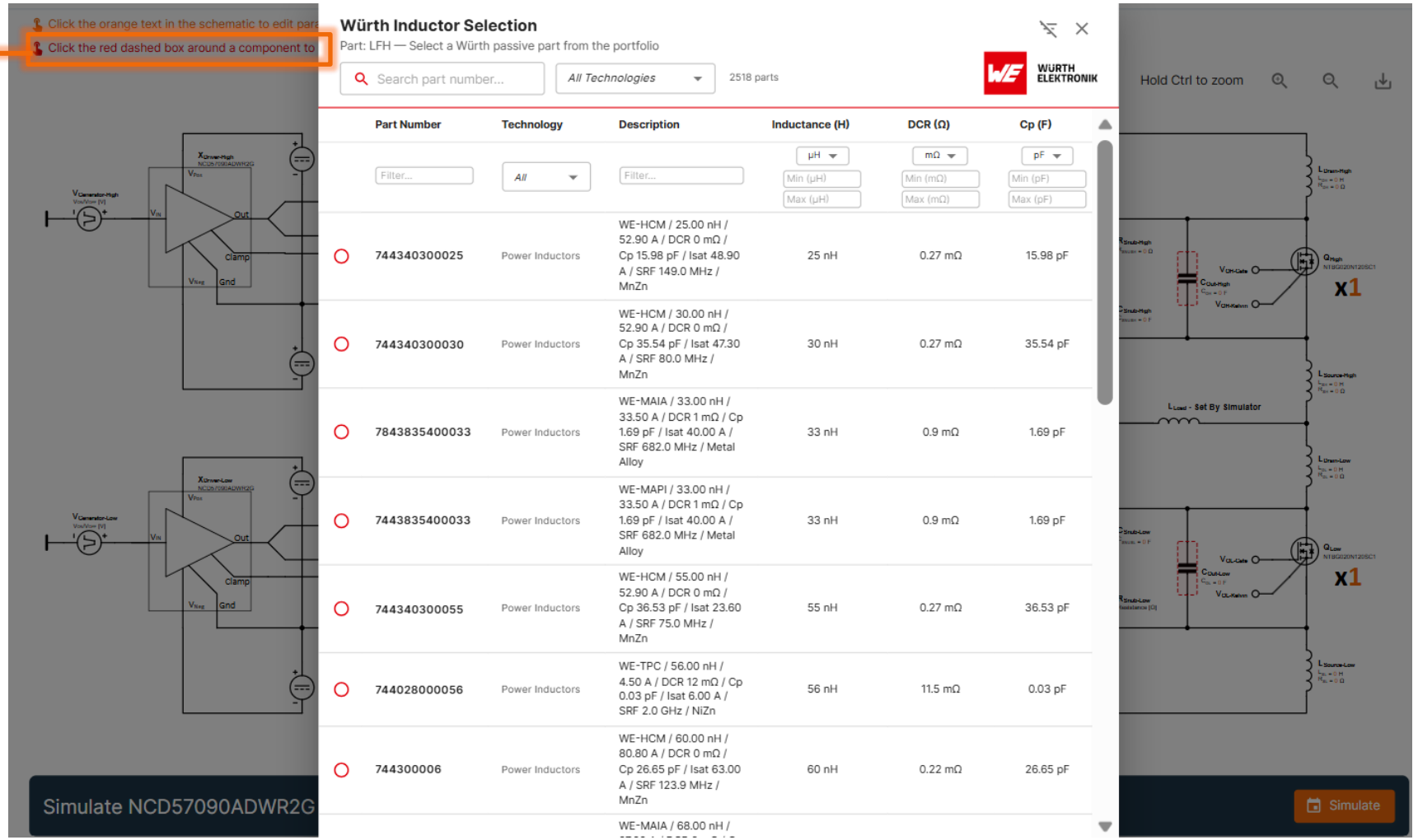
Diode model can be selected from onsemi's portfolio



After clicking the diode, a selection window opens

Step 2: Setting components and parasitic elements (cont.)

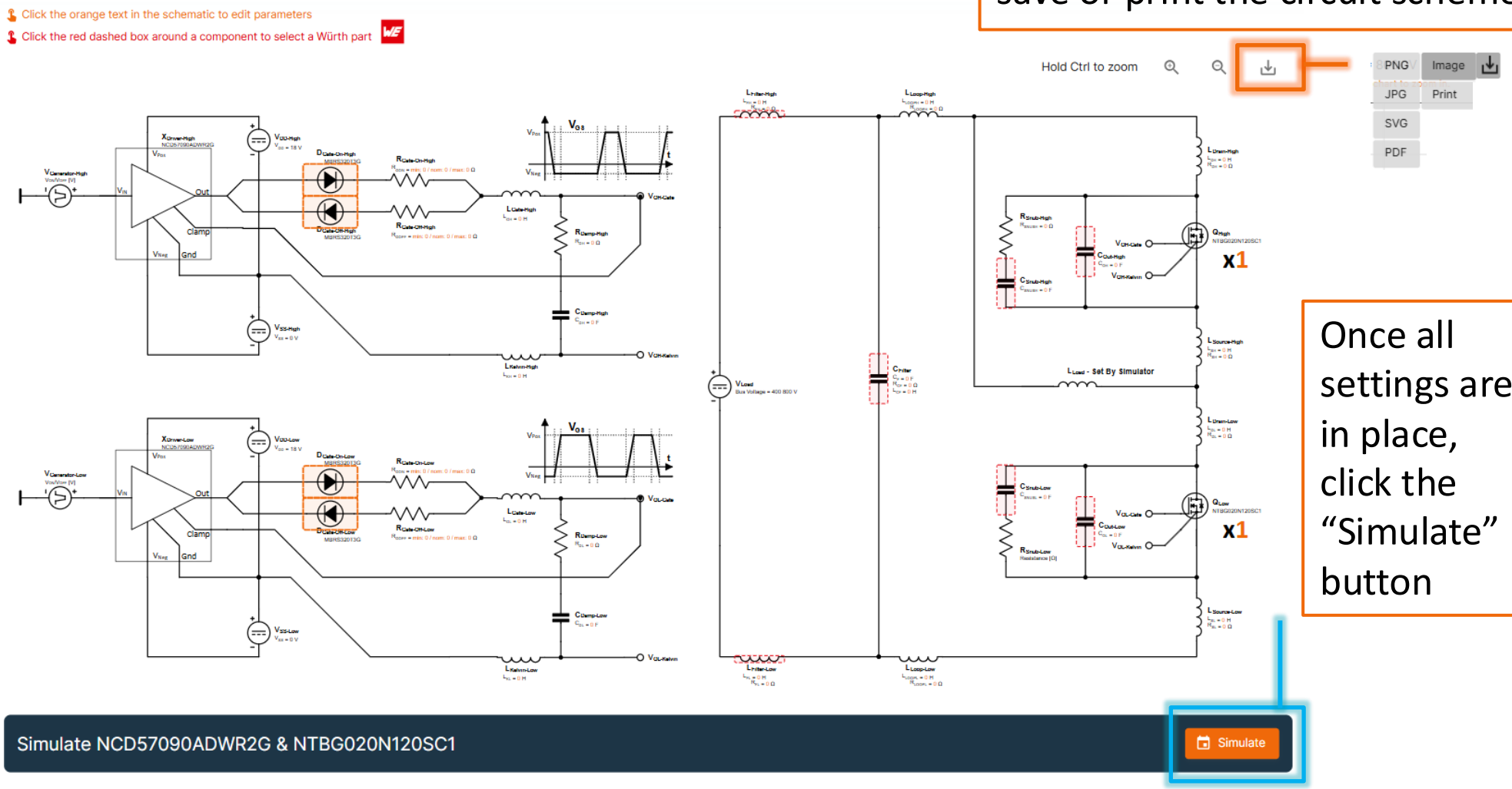
Passive components can be selected from Wurth portfolio



After clicking the passive component, a selection window opens

Step 3: Running the SPICE simulations

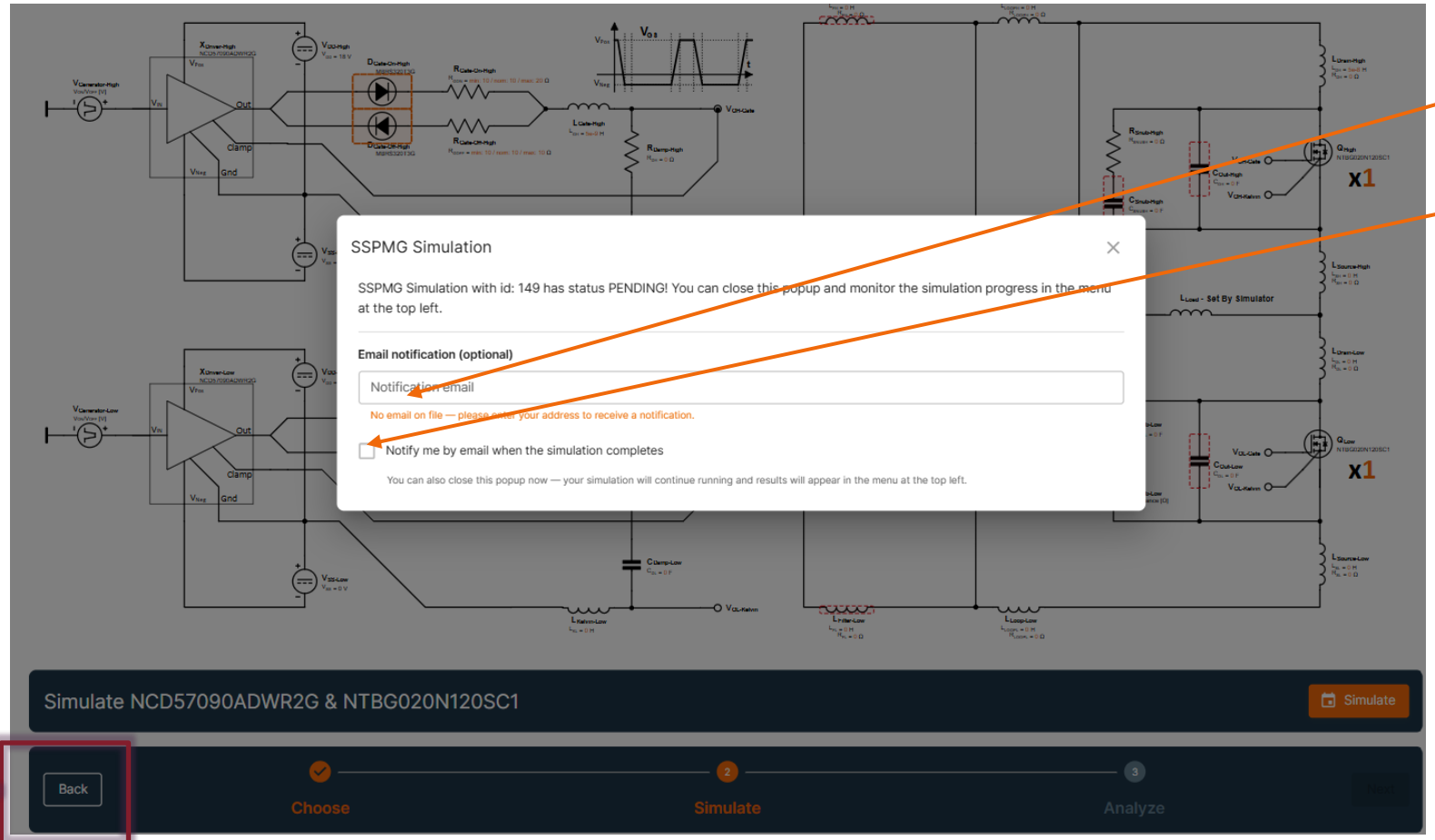
Click this button if you want to save or print the circuit scheme



Once all settings are in place, click the “Simulate” button

Step 3: Running the SPICE simulations (cont.)

After clicking “Simulation”, a window opens with the option to send you an e-mail alerting that simulations are finished and ready for “Analyze”



Type your e-mail address and check the “Notify me by mail” box

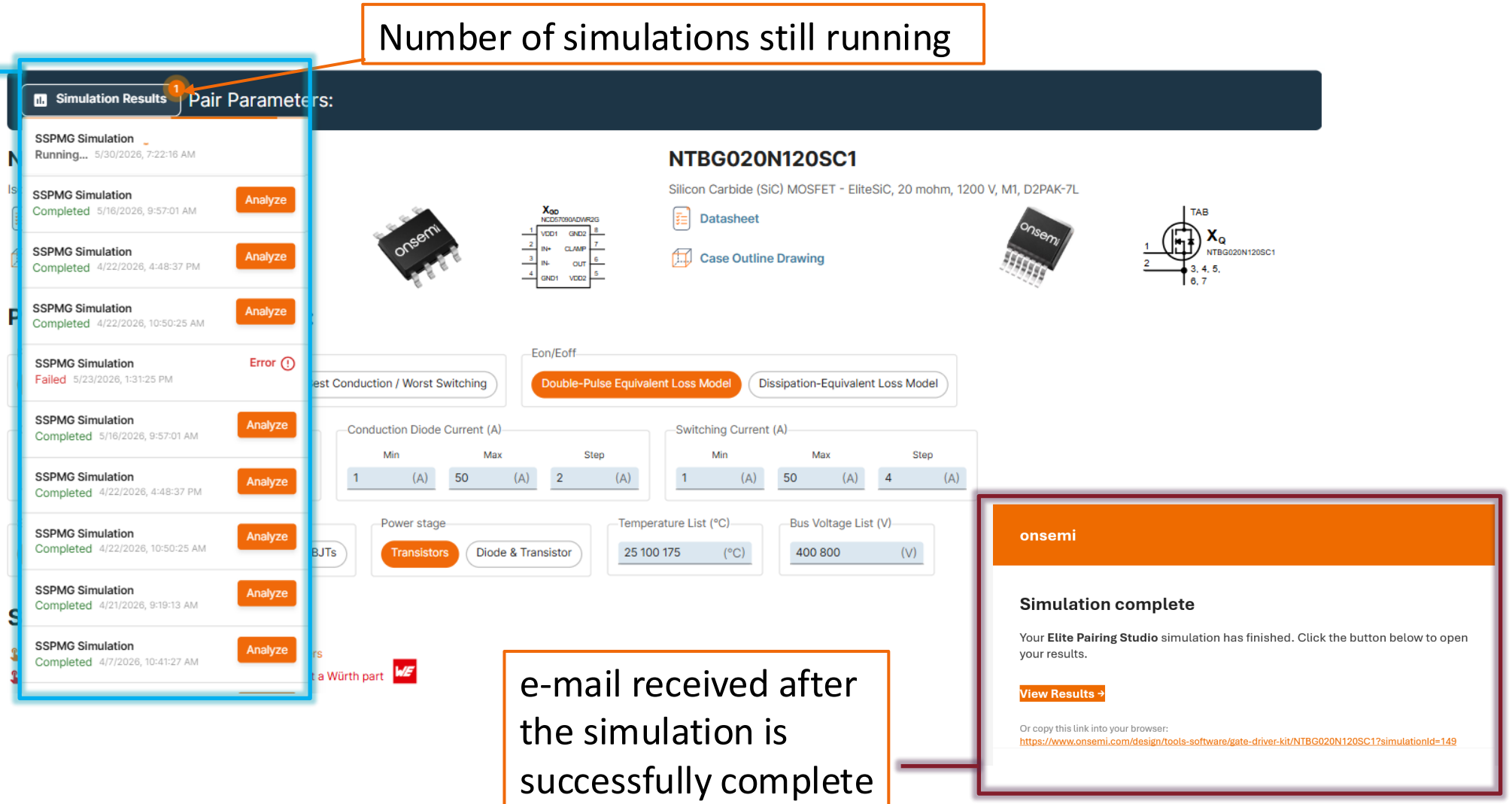
You can return at Stage 1 at any moment

This option is useful when simulations take a long time

Step 3: Running the SPICE simulations (cont.)

List and status of simulations

Number of simulations still running

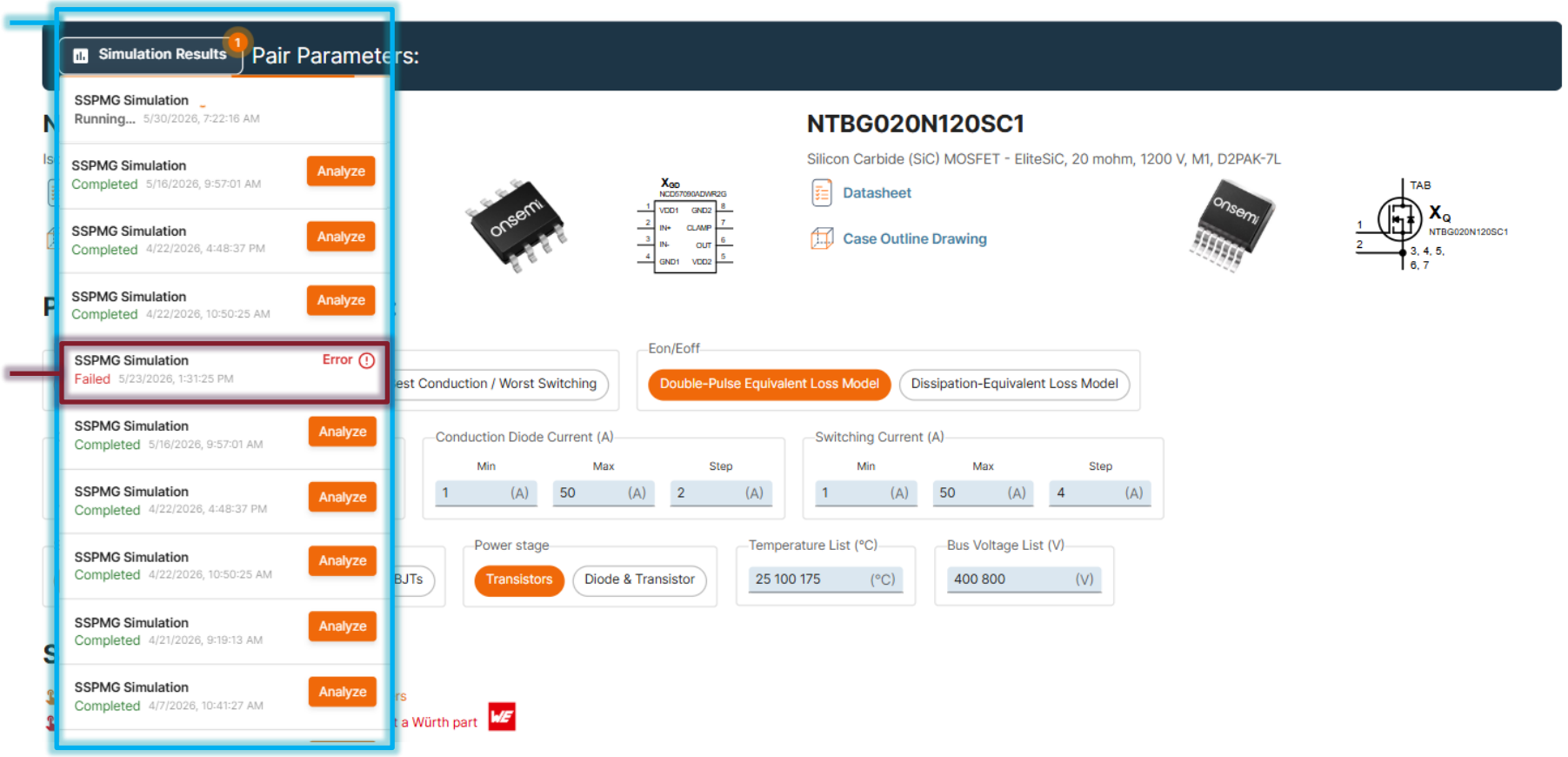


e-mail received after the simulation is successfully complete

Step 3: Running the SPICE simulations (cont.)

List and status of simulations

In case of “Failed” simulation, a mail is also sent to the user, and our Support team is notified to investigate the issue

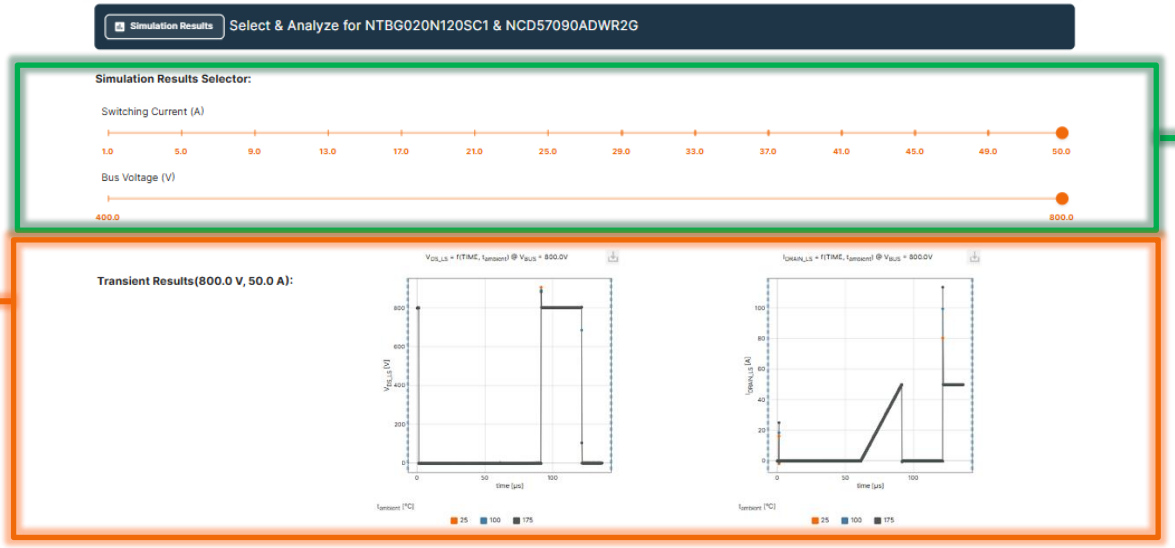


Outline of User Guide

1	Introduction to Elite Pairing Studio
2	Stage 1: Choose
3	Stage 2: Simulate
4	Stage 3: Analyze

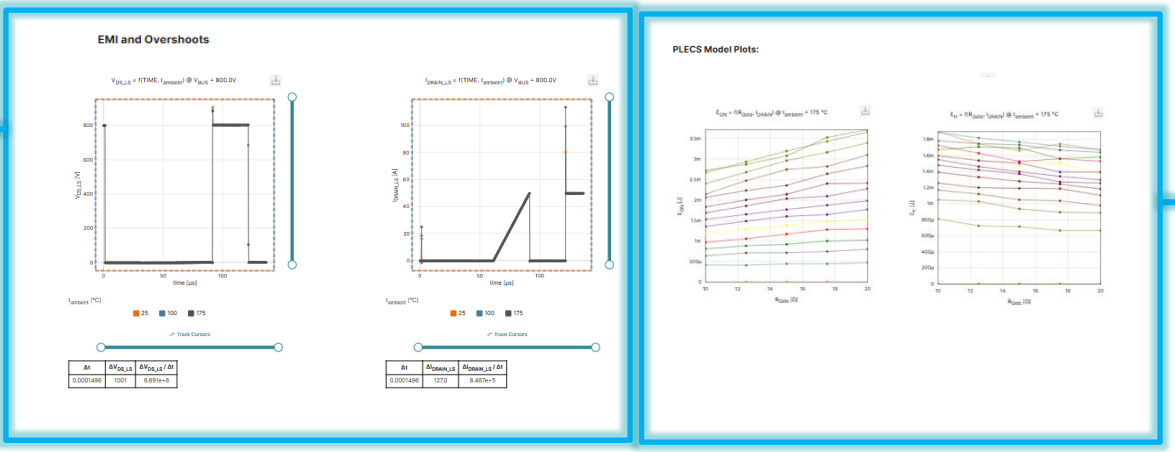
General View

Plots and parametric analysis of simulated waveforms



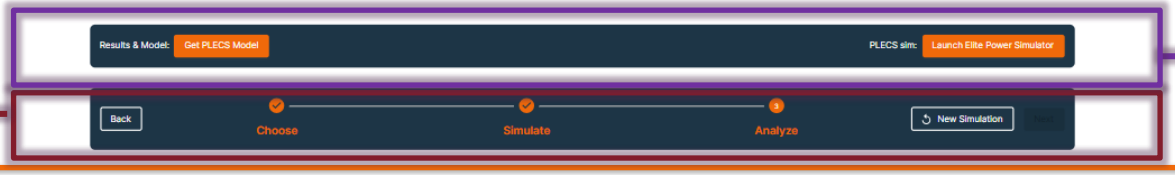
Filter for range of switching current and bus voltage

Plots and parametric analysis of current and voltage overshoots



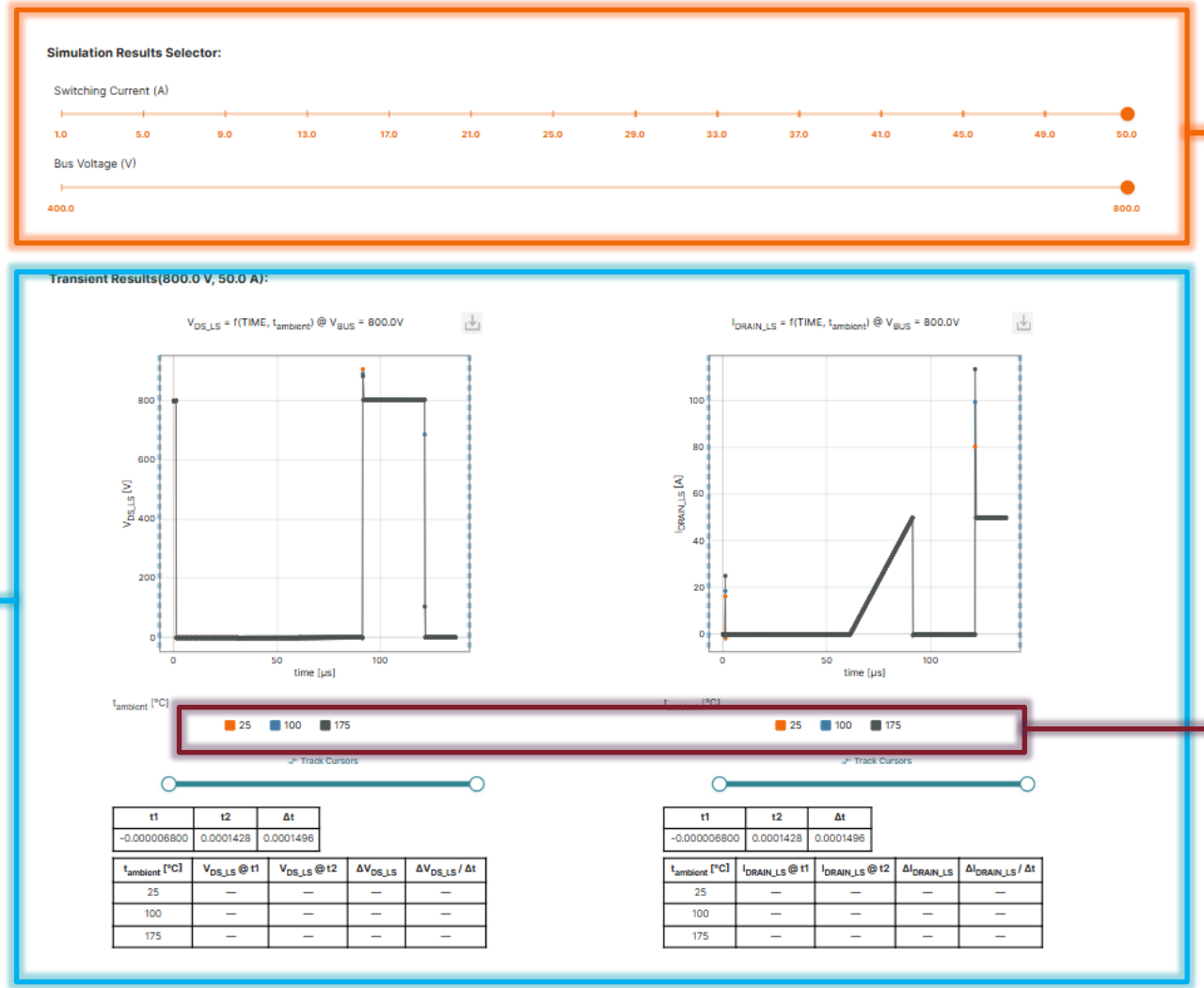
Plots and parametric analysis of switching energies and IV curves (also used to generate a PLECS model)

Flowline between Stages



Links to PLECS model generator and Elite Power Simulator

Step 1: Using filters to display transient results



Select a single value for switching current and bus voltage

Plots with simulated waveforms are updated according to the Simulation Results Selector

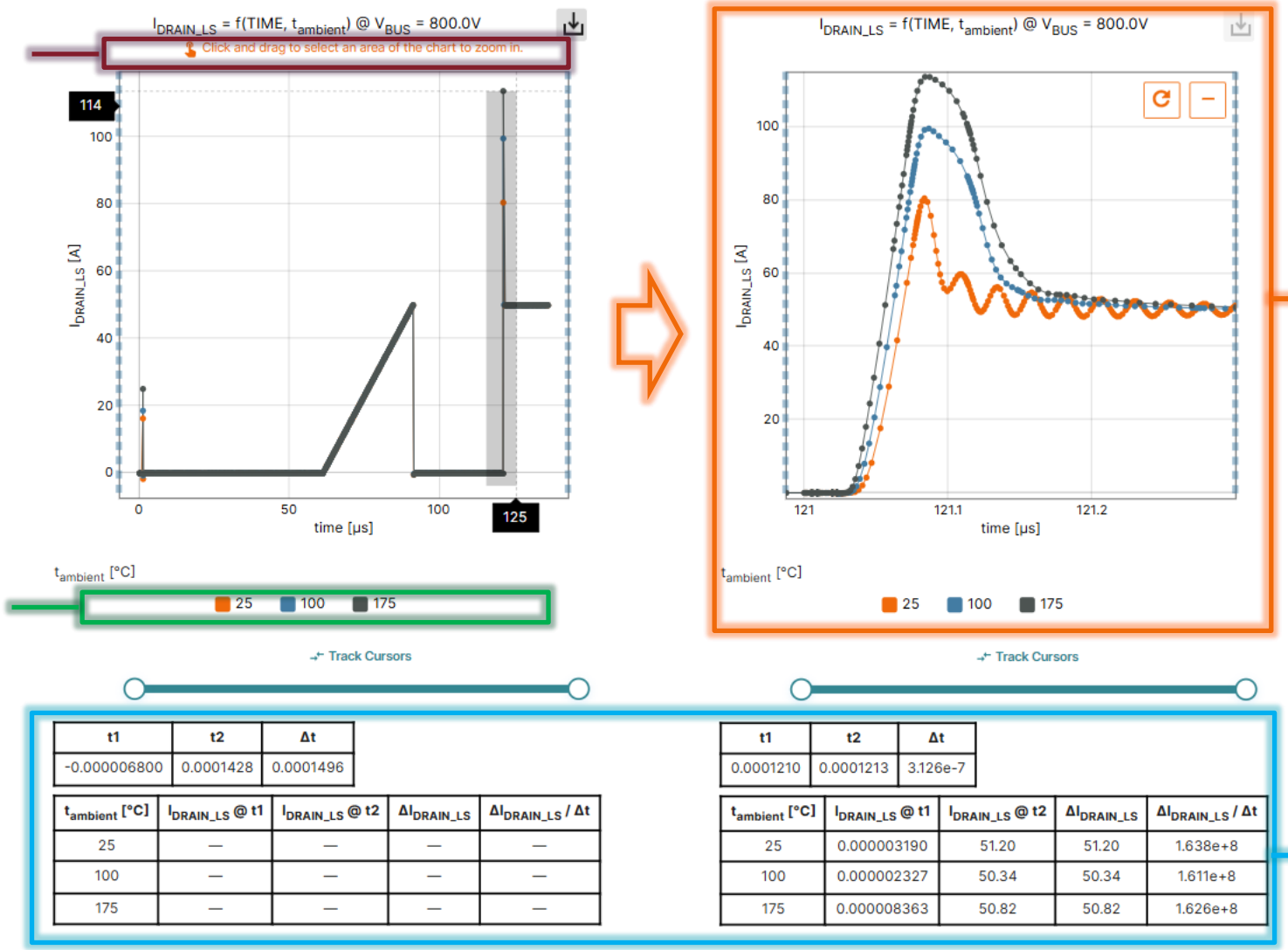
By scrolling down, 5 plots appear representing V_{DS_LS}, I_{DRAIN_LS}, V_{GS_LS}, V_{DS_HS} and I_{DRAIN_HS}

Every plot displays as many waveforms as temperatures were defined in Stage 2

Step 2: Interacting with the transient plots

Click and drag to select an area of the chart to zoom in

Click the legend boxes to display or hide waveforms



t1	t2	Δt
-0.000006800	0.0001428	0.0001496

$t_{ambient}$ [°C]	I_{DRAIN_LS} @ t1	I_{DRAIN_LS} @ t2	ΔI_{DRAIN_LS}	ΔI_{DRAIN_LS} / Δt
25	—	—	—	—
100	—	—	—	—
175	—	—	—	—

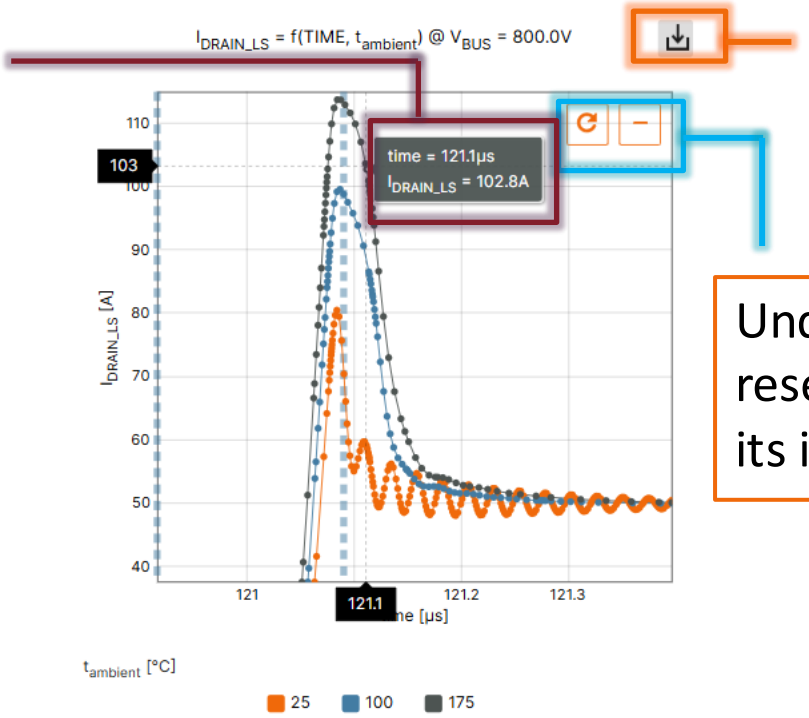
t1	t2	Δt
0.0001210	0.0001213	3.126e-7

$t_{ambient}$ [°C]	I_{DRAIN_LS} @ t1	I_{DRAIN_LS} @ t2	ΔI_{DRAIN_LS}	ΔI_{DRAIN_LS} / Δt
25	0.000003190	51.20	51.20	1.638e+8
100	0.000002327	50.34	50.34	1.611e+8
175	0.000008363	50.82	50.82	1.626e+8

Table with data on t1, t2 times, current or voltage at t1, t2 and t1-t2 difference

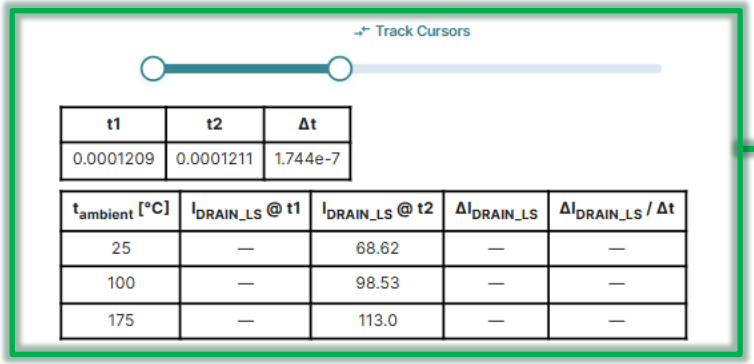
Step 2: Interacting with the transient plots (cont.)

Place your pointer to obtain the plot coordinates at a given waveform point



Click this button if you want to save or print the waveforms

Undo zoom or reset the plot to its initial state



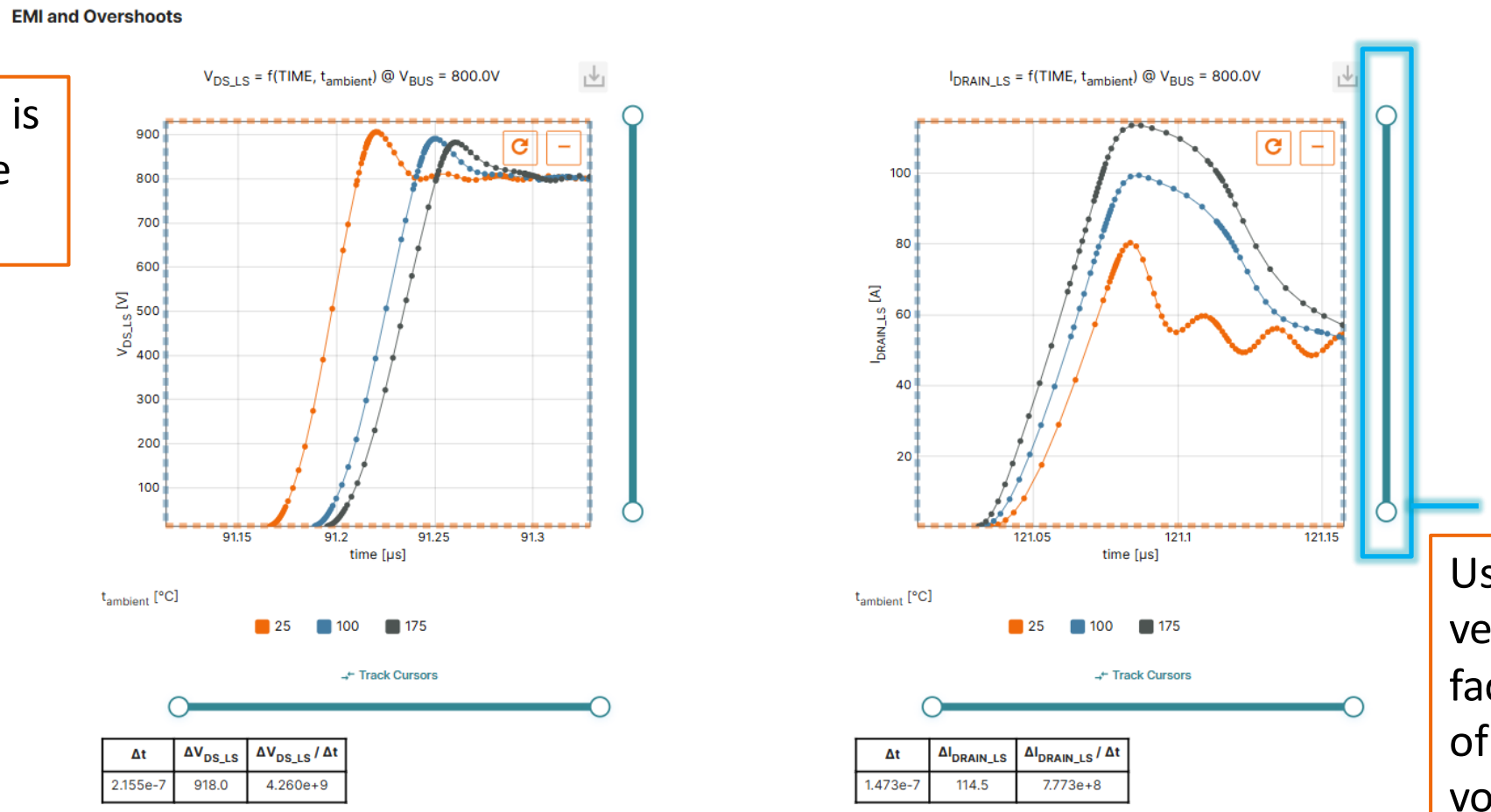
t1	t2	Δt
0.0001209	0.0001211	1.744e-7

$t_{ambient}$ [°C]	I_{DRAIN_LS} @ t1	I_{DRAIN_LS} @ t2	ΔI_{DRAIN_LS}	ΔI_{DRAIN_LS} / Δt
25	—	68.62	—	—
100	—	98.53	—	—
175	—	113.0	—	—

Drag the track cursors to extract the current or voltage values at a given time

Step 3: Interacting with the overshoot plots

The interaction is similar as in the transient plots



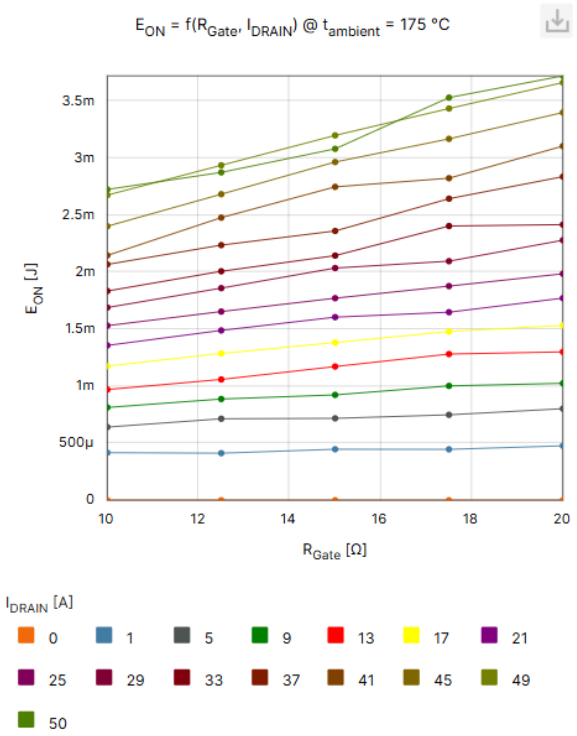
Use the additional vertical track bar to facilitate calculation of current and voltage overshoots

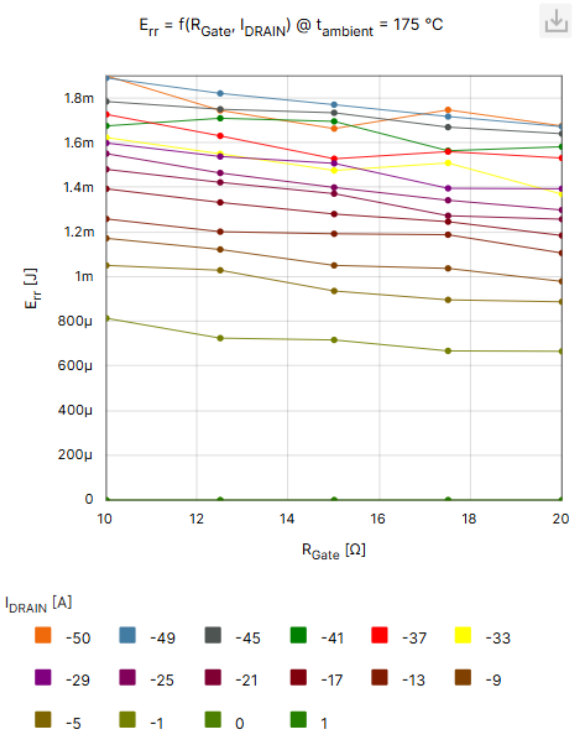
Step 4: Interacting with the PLECS model plots

The interaction is similar as in the transient and overshoot plots

Three plots for E_{ON}, E_{OFF} and E_{RR} vs. R_{Gate} are displayed using I_{DRAIN} as a parameter (values selected in Stage 2)

PLECS Model Plots:

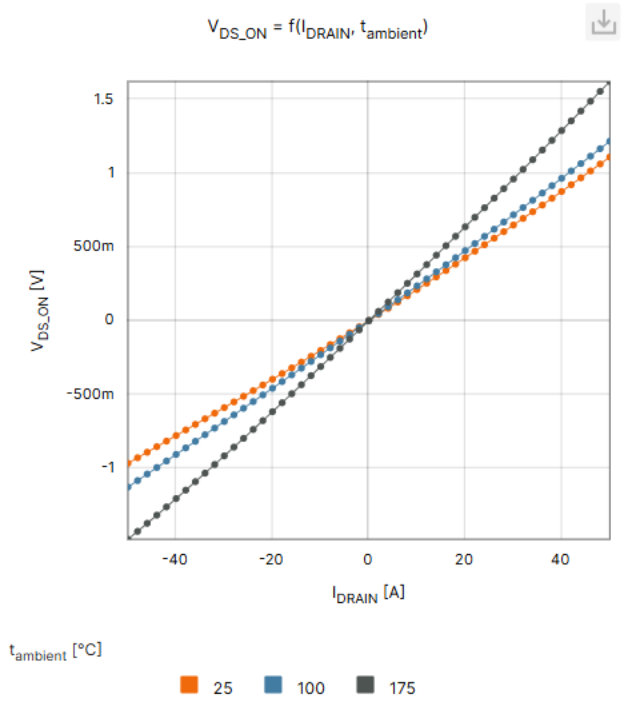


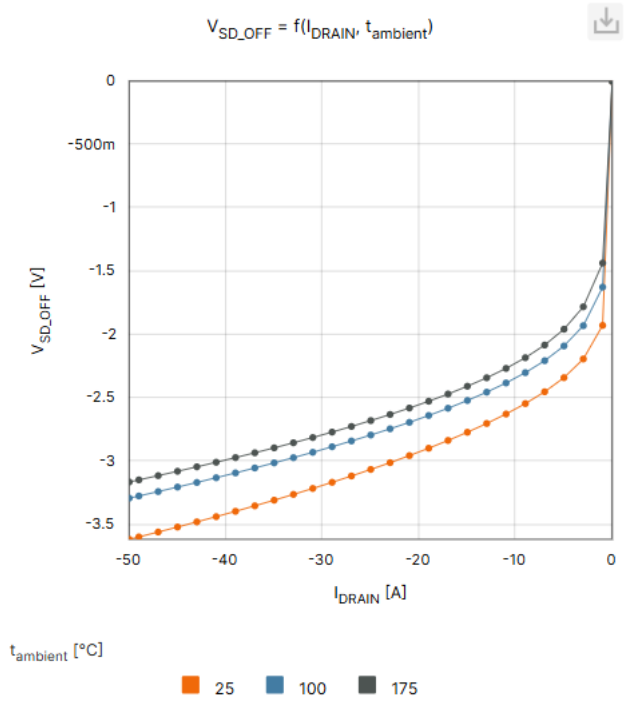


By scrolling down, two additional plots for E_{ON} and E_{OFF} vs. I_{DRAIN} are displayed using temperature and bus voltage as parameters (typical R_{Gate})

Step 4: Interacting with the PLECS model plots (cont.)

The interaction is similar as in the transient and overshoot plots





By scrolling down, two plots for V_{DS_ON} and V_{SD_OFF} vs. I_{DRAIN} are displayed using temperature as a parameter (values selected in Stage 2)

Step 5: Downloading the PLECS model

Scroll down until the bottom of the webpage

Click the “Get PLECS Model” button to download the generated PLECS models

Additional simulation options are offered by Elite Power Simulator

You can return at Stage 2 at any moment

Start a new fresh simulation

Questions?

Have questions, comments, or need support with your Elite Pairing Studio? We're here to help! Write us an email at **elitepair@onsemi.com**

- Elite Pairing Studio:

  www.onsemi.com/design/elite-pairing-studio

onsemi™

Intelligent Technology. Better Future.

Follow us **@onsemi**

onsemi.com