



# Elite Pairing Studio

## User Guide

Ver. 1.0

© onsemi 2026 | Public Information



# Outline of User Guide

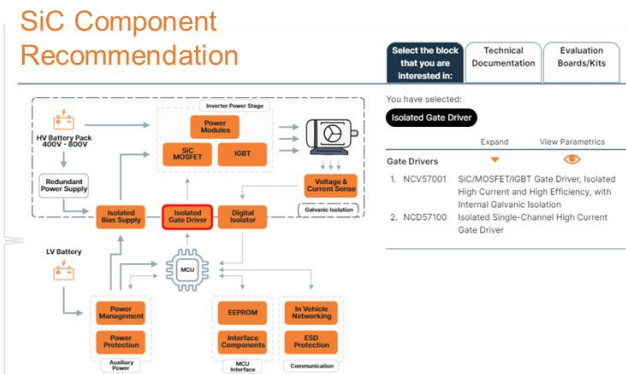
1	Introduction to Elite Pairing Studio
2	Stage 1: Choose
3	Stage 2: Simulate
4	Stage 3: Analyze

# 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.



## Product Filtering Tool

**Quick Filters**

Power Switch: **SiC MOSFET**

Qualification: **All** | Automotive | Standard

Isolation Type: **All**

Topology: **All**

Number of Outputs: **All**

Package Type: **All**

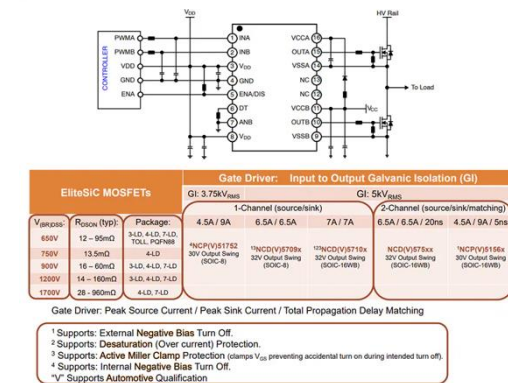
Drive Source Current  $I_{PK(SW)}$  Typ (A)  $\geq 0.0265$  A  $\leq 11.4$  A

Drive Sink Current  $I_{PK(SW)}$  Typ (A)  $\geq 0.0265$  A  $\leq 10.6$  A

Voltage Class  $V_{CLASS, Max}$  (V)  $\geq -518$  V  $\leq 1225$  V

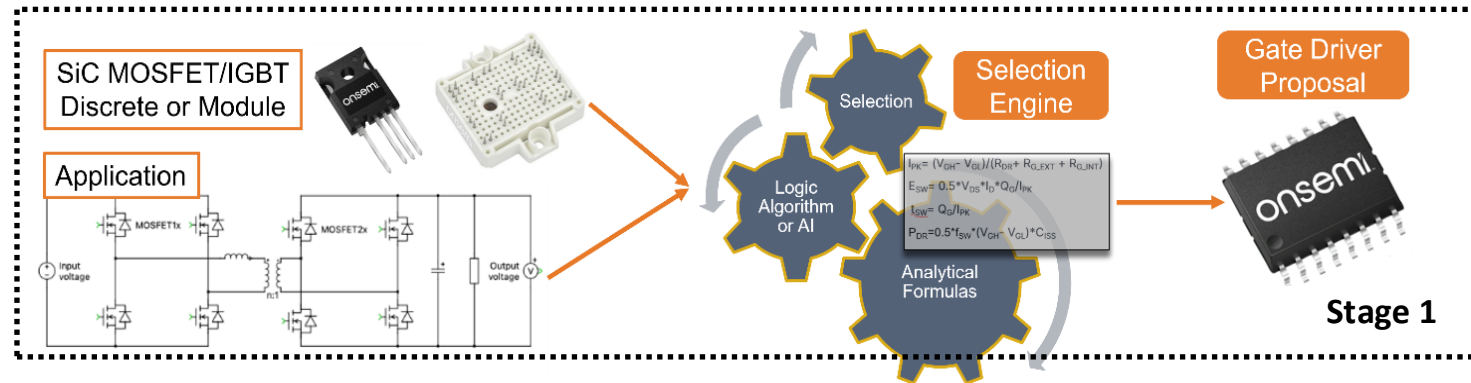
Supply voltage  $V_{CC}$  Max (V)  $\geq 5.25$  V

EliteSiC to Gate Driver Pairing



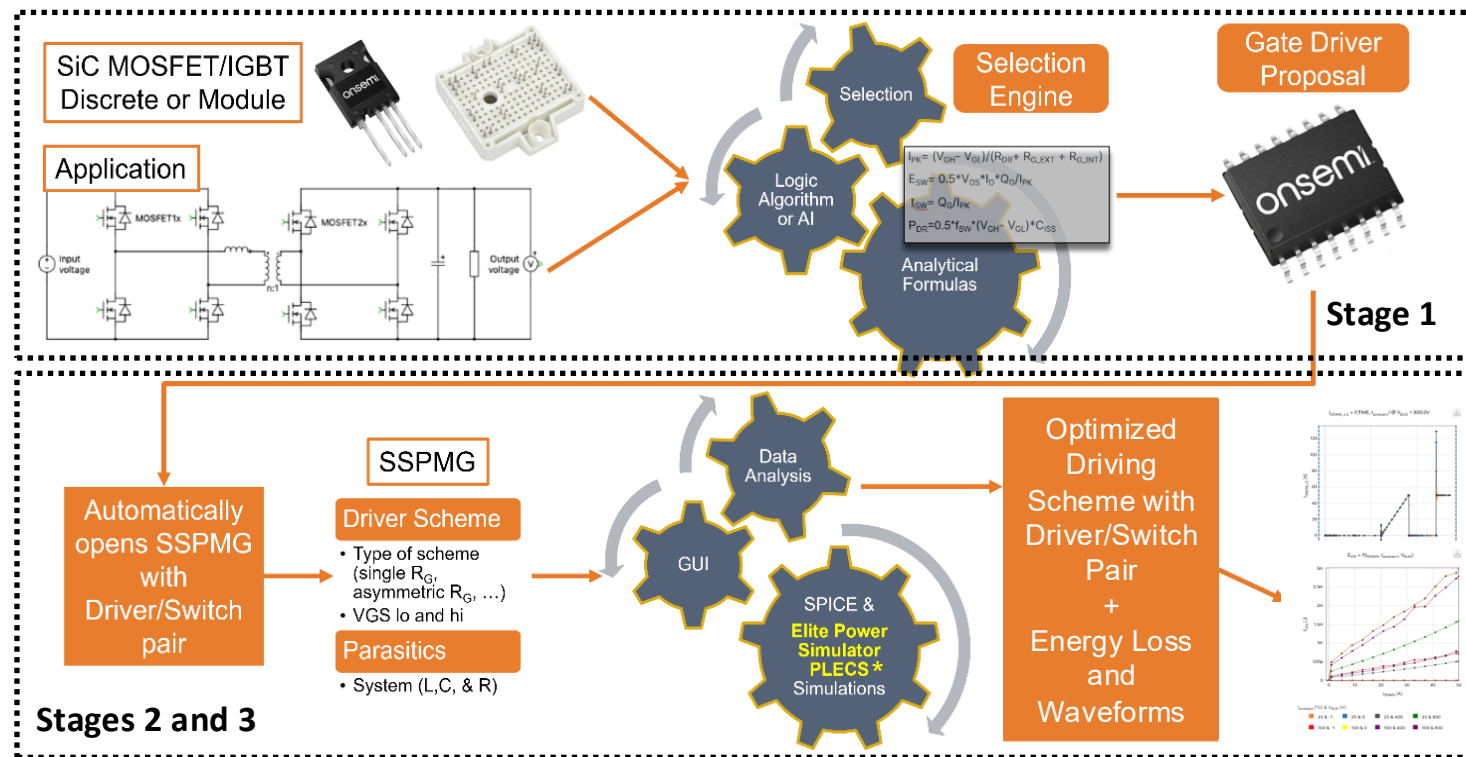
# Introduction

- 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.



# Introduction

## Access Elite Pairing Studio with MYON Account

### MYON Login

onsemi Products Solutions Design Support Company Careers

Search the Site & Cross Reference

MYON Login

### Unlock Awesome Perks When you Sign up!

- Order product samples & evaluation boards
- Save custom parametric search filters
- Join the conversation on community forum
- Supercharge your projects with the Elite Power Simulator and design tools.
- Keep your block diagrams worksheets handy—save or email them anytime.
- Catch exclusive webinars and seminars you won't find anywhere else.
- Easily revisit your favorites with My History browsing.
- Dive into our system solution guides for expert insights.

### Login

**Existing User**

Email

Password

Forgot Password

Login

Do not have an account? **Register**

**First Time User**

# Outline of User Guide

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

# General View

Selected SiC MOSFET and corresponding information

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

Flowline between Stages

The screenshot shows the 'Elite Pairing Studio' interface for the NTBG020N120SC1 MOSFET. It includes a search bar, navigation links (Datasheet, Product Page, Find Similar Products, Case Outline Drawing), and a schematic diagram of the MOSFET. Below the schematic, there are sections for 'Gate Driver Filters', 'MOSFET Driving Conditions' (with a waveform showing  $V_{DR}$  and  $V_{GS}$ ), and a 'Gate Source Voltage vs. Total Charge' graph. At the bottom, there is a 'Choose Suitable Gate Driver' table with columns for various parameters and a progress indicator with steps: Choose, Simulate, Analyze.

OPN	$t_{th}$ (ns)	$I_{pk}$ (A)	$t_{tr}$ (ns)	$t_{on}$ (ns)	Initial Mode	$E_{cond}$ ( $\mu$ J)	$E_{th}$ ( $\mu$ J)	$E_{class}$ ( $\mu$ J)	$T_j$ ( $^{\circ}$ C)	$T_j$ Max ( $^{\circ}$ C)	Price (\$)	$I_{pk\_SIC\_Typ}$ (A)	$I_{vc\_SIC\_Typ}$ (A)	$V_{GS\_Min}$ (V)
<input type="checkbox"/> 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
<input type="checkbox"/> 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
<input type="checkbox"/> 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
<input type="checkbox"/> 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
<input type="checkbox"/> NCD57090DWR2G	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

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"

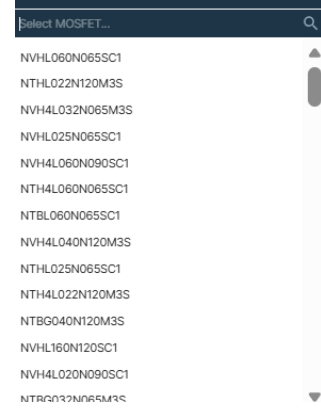
Gate charge curve for selected SiC MOSFET (extracted from SPICE model)

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



A screenshot of the onsemi website interface. At the top, there is a navigation bar with a "Simulation Results" button (highlighted in green), a search bar "Looking for a MOSFET?" with a "Select MOSFET..." dropdown (highlighted in orange), and a link "Explore MOSFET Portfolio: Product Recommendation Tools+™" (highlighted in red). Below the navigation bar, the main content area shows "Elite Pairing Studio\_beta | NTB020N120SC1" with a description "Silicon Carbide (SiC) MOSFET - EliteSiC, 20 mohm, 1200 V, M1, D2PAK-7L". There are links for "Datasheet", "Product Page", "Find Similar Products", "Case Outline Drawing", "User Guide", "App Note", and "Feedback". To the right, there is a "Pinmap:" section with a schematic diagram of the MOSFET and a "Package 3D:" section with a 3D model of the component.

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

The screenshot shows the onsemi design tool interface for the NTBG020N120SC1 MOSFET. It is divided into three main sections:

- Gate Driver Filters:** A sidebar on the left with dropdown menus for Topology / Configuration (All), Isolation Type (All), Qualification (All, Automotive, Standard), Number of Outputs (All), and Features / Spec. Functions.
- MOSFET Driving Conditions:** A central schematic diagram of the gate driver circuit. It includes an ISO block, a driver IC, and the MOSFET. Parameters shown include  $V_{CC}$ ,  $V_{EE}$ ,  $V_{DR}$ ,  $R_{GEXT} = 2.5 \Omega$ ,  $R_{GINT} = 1.85 \Omega$ ,  $n_{DIE} = 1$ , and  $t_{ambient} = 25^\circ C$ . A waveform graph above the schematic shows a trapezoidal pulse with  $f_{switch} = 50 \text{ kHz}$  and  $T = \frac{1}{f_{switch}}$ . A note says "Click the orange text in the schematic to edit parameters".
- Gate Source Voltage vs. Total Charge:** A graph on the right showing  $V_{GS}$  (V) on the y-axis (ranging from -10 to 25) versus  $Q_G$  (nC) on the x-axis (ranging from 0 to 350). A blue curve shows the gate voltage profile. A point on the curve is highlighted with an orange box and labeled "Q<sub>G</sub> Drive = 209.36 (nC)". A note above the graph states: "Note: stated curve is generated at different condition compared to datasheet."

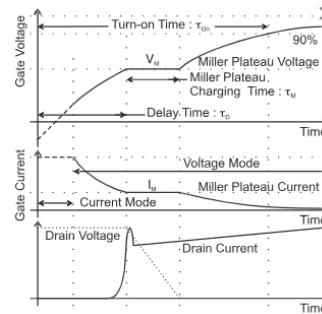
At the bottom of the interface, there are two sliders for "Supply Voltage  $V_{CC}$  Max (V)" and "Supply Voltage  $V_{EE}$  Min (V)". The  $V_{CC}$  slider is set to  $\geq 20$  V and the  $V_{EE}$  slider is set to  $\leq 0$  V.

$Q_G$  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  $\tau_M$ ,  $I_M$ ,  $\tau_D$  and  $\tau_{On}$



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](#)

	$\tau_M$ (ns) ↑	$I_M$ (A)	$\tau_D$ (ns)	$\tau_{On}$ (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)
<input type="checkbox"/> 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
<input type="checkbox"/> 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
<input type="checkbox"/> 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
<input type="checkbox"/> 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
<input type="checkbox"/> 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

1 Choose      2 Simulate      3 Analyze

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).

### Choose Suitable Gate Driver

Note: simulation is supported for a subset of gate drivers

[Description of variables in the table](#)

Columns

OPN	$t_M$ (ns)	$I_M$ (A)	$t_D$ (ns)	$t_{On}$ (ns)	Initial Mode	$E_{GD}$ ( $\mu$ J)	$E_{Rg}$ ( $\mu$ J)	$E_{CISS}$ ( $\mu$ J)	$T_J$ ( $^{\circ}$ C)	$T_J$ Max ( $^{\circ}$ C)	Price (\$)	$I_{PK\ SNK}$ Typ (A)	$I_{PK\ SRC}$ Typ (A)	$V_{EE}$ Min (V)
<input checked="" type="checkbox"/> 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
<input type="checkbox"/> 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
<input type="checkbox"/> 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
<input type="checkbox"/> 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
<input type="checkbox"/> 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

1 row selected

Rows per page: 5 1-5 of 38

Back 1 Choose 2 Simulate 3 Analyze Next

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

The screenshot displays the onsemi simulation tool interface. At the top, a dark blue bar contains 'Simulation Results' and 'Pair Parameters:'. Below this, two components are selected: NCD57090ADWR2G (Isolated High Current Gate Driver) and NTB020N120SC1 (Silicon Carbide (SiC) MOSFET - EliteSiC, 20 mohm, 1200 V, M1, D2PAK-7L). Each component has links for 'Datasheet' and 'Case Outline Drawing' and a small image of the component. Below the component selection is the 'Pair Simulation Parameters:' section, which includes 'Process Conditions' (Nominal, Worst Conduction / Best Switching, Best Conduction / Worst Switching, Double-Pulse Equivalent Loss Model, Dissipation-Equivalent Loss Model), 'Conduction Transistor Current (A)', 'Conduction Diode Current (A)', and 'Switching Current (A)' with min, max, and step values. It also includes 'Switching Type' (Hard, Soft, Diodes, BJT & Diode, BJTs, Transistors, Diode & Transistor), 'Gate Driver with', 'Power stage', 'Temperature List (°C)', and 'Bus Voltage List (V)'. The 'Schematic:' section shows two circuit diagrams. The left diagram is a gate driver circuit with a MOSFET and a diode, and the right diagram is a power stage circuit with a MOSFET, diode, and inductor. Below the schematics is a 'Simulate NCD57090ADWR2G & NTB020N120SC1' button with a 'Simulate' icon. At the bottom, a navigation bar shows 'Back', 'Choose', 'Simulate', and 'Analyze' steps.

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

**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

Conduction Transistor Current (A):  
Min: -50 (A) Max: 50 (A) Step: 2 (A)

Conduction Diode Current (A):  
Min: 1 (A) Max: 50 (A) Step: 2 (A)

Switching Current (A):  
Min: 1 (A) Max: 50 (A) Step: 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)

“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\*

**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

Conduction Transistor Current (A): Min: -50 (A), Max: 50 (A), Step: 2 (A)

Conduction Diode Current (A): Min: 1 (A), Max: 50 (A), Step: 2 (A)

Switching Current (A): Min: 1 (A), Max: 50 (A), Step: 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

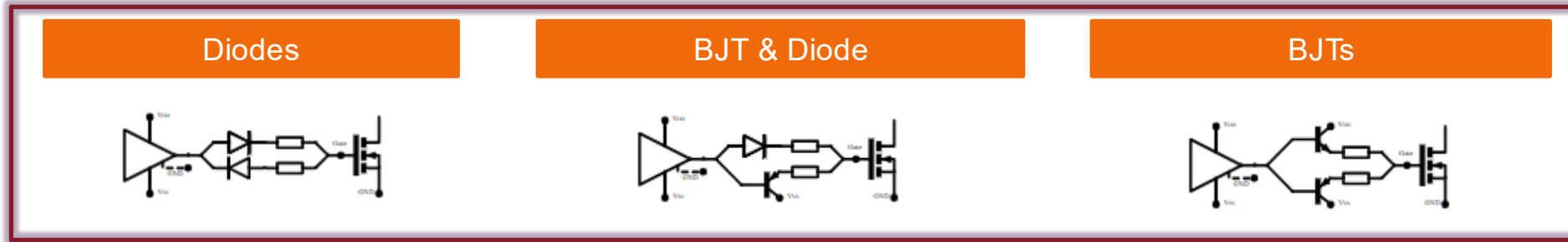
Delay between transistors (ns): Max Delay (ns)

“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

Conduction Transistor Current (A): Min: -50 (A), Max: 50 (A), Step: 2 (A)

Conduction Diode Current (A): Min: 1 (A), Max: 50 (A), Step: 2 (A)

Switching Current (A): Min: 1 (A), Max: 50 (A), Step: 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

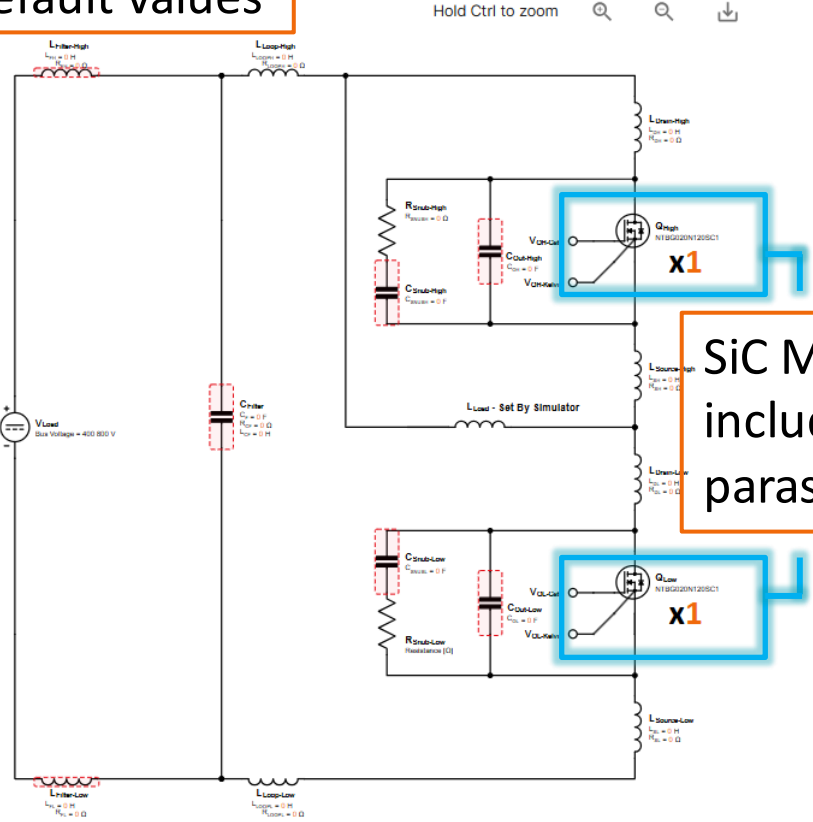
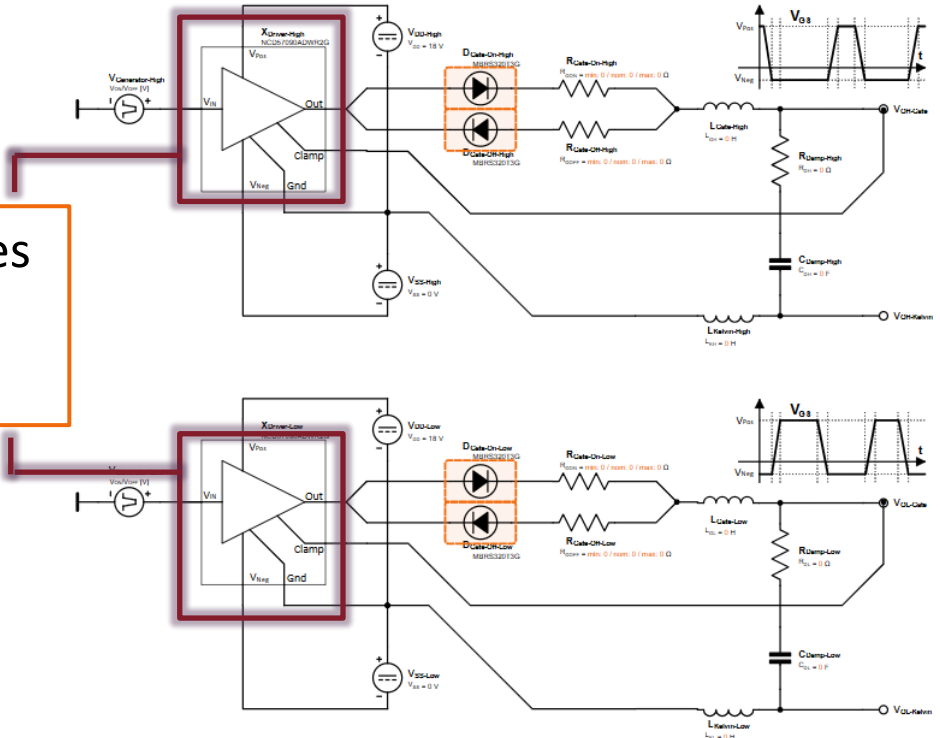
# Step 2: Setting components and parasitic elements

Default value for capacitances, inductances and resistances is zero

Click the orange text in the schematic to edit parameters  
 Click the red dashed box around a component to select a Würth part

Click the orange text to change the default values

GD model includes package parasitic elements



SiC MOSFET model includes package parasitic elements

Simulate NCD57090ADWR2G & NTB020N120SC1 Simulate

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

## Conventional zooming techniques

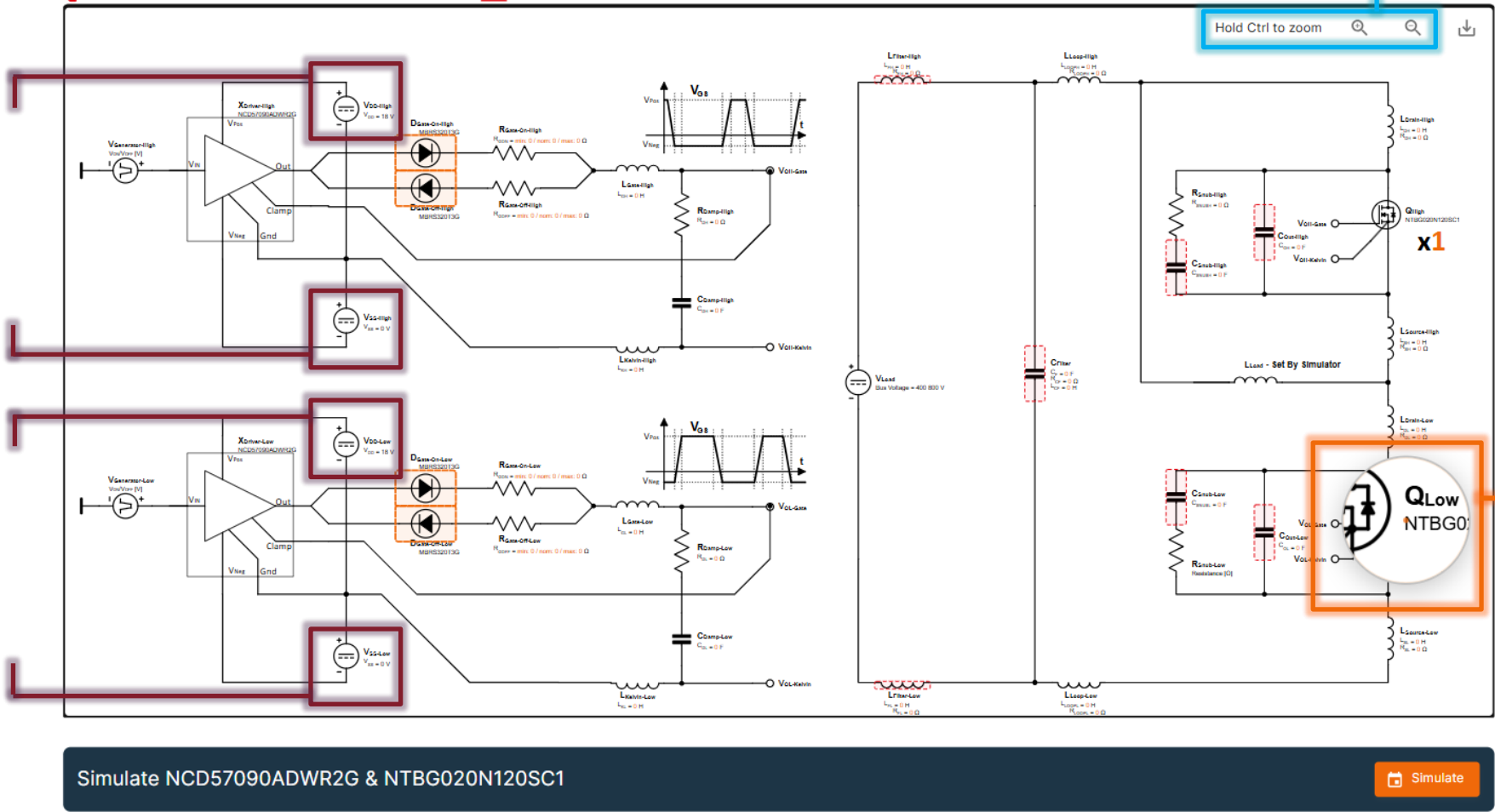
Click the orange text in the schematic to edit parameters

Click the red dashed box around a component to select a Würth part



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

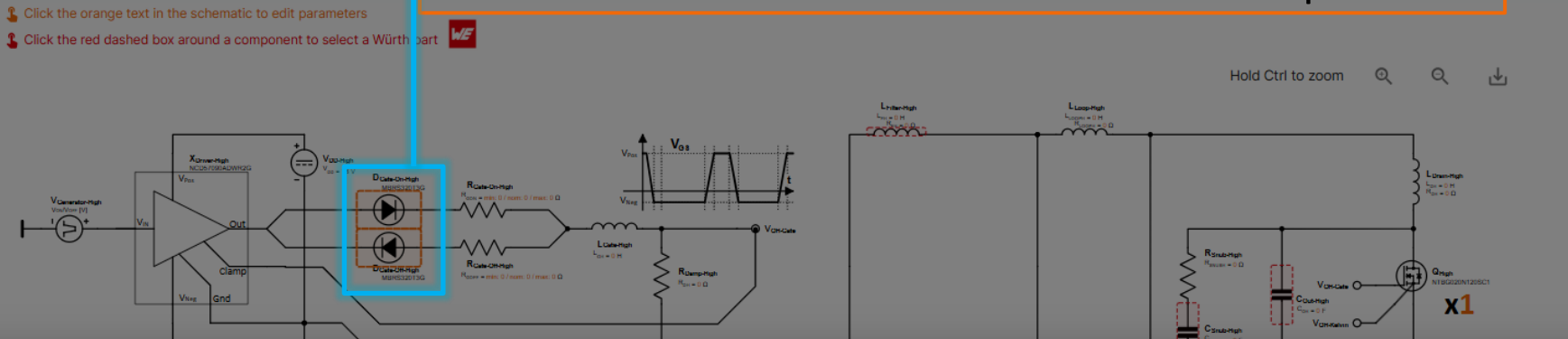
Click the orange text in the schematic to edit parameters  
Click the red dashed box around a component to select a Würth part

Simulate NCD57090ADWR2G & NTB020N120SC1

Simulate

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

Diode model can be selected from onsemi's portfolio



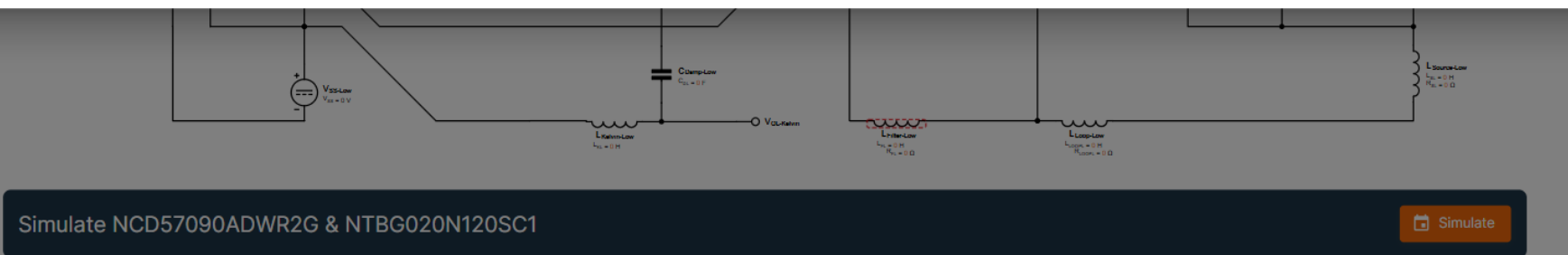
After clicking the diode, a selection window opens

## Output Stage Part Selection

Diode + Diode pair — select a pair to use in the simulation

Gate-ON Part — Diode								Gate-OFF Part — Diode				
OPN	Type	Package	VRRM (V)	IF(ave) (A)	IFSM (A)	VF (V)	IR (μA)	OPN	Type	Package	VRRM (V)	IF(ave) (A)
<input type="text" value="Filter..."/>		<input type="text" value="Filter..."/>	Min Max	Min Max	Min Max	Min Max	Min Max	<input type="text" value="Filter..."/>		<input type="text" value="Filter..."/>	Min Max	Min Max
<input checked="" type="radio"/> MBRS320T3G	Diode	SMC-2	20	3	80	0.5	2000	MBRS320T3G	Diode	SMC-2	20	3
<input type="radio"/> MBRS540T3G	Diode	SMC-2	40	5	190	0.5	300	MBRS540T3G	Diode	SMC-2	40	5

Selected: MBRS320T3G + MBRS320T3G



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

Passive components can be selected from Würth portfolio

Click the orange text in the schematic to edit parasitic elements  
Click the red dashed box around a component to select it

### Würth Inductor Selection

Part: LFH — Select a Würth passive part from the portfolio

Search part number... All Technologies 2518 parts

Part Number	Technology	Description	Inductance (H)	DCR (Ω)	Cp (F)
744340300025	Power Inductors	WE-HCM / 25.00 nH / 52.90 A / DCR 0 mΩ / Cp 15.98 pF / Isat 48.90 A / SRF 149.0 MHz / MnZn	25 nH	0.27 mΩ	15.98 pF
744340300030	Power Inductors	WE-HCM / 30.00 nH / 52.90 A / DCR 0 mΩ / Cp 35.54 pF / Isat 47.30 A / SRF 80.0 MHz / MnZn	30 nH	0.27 mΩ	35.54 pF
7843835400033	Power Inductors	WE-MAIA / 33.00 nH / 33.50 A / DCR 1 mΩ / Cp 1.69 pF / Isat 40.00 A / SRF 682.0 MHz / Metal Alloy	33 nH	0.9 mΩ	1.69 pF
7443835400033	Power Inductors	WE-MAPI / 33.00 nH / 33.50 A / DCR 1 mΩ / Cp 1.69 pF / Isat 40.00 A / SRF 682.0 MHz / Metal Alloy	33 nH	0.9 mΩ	1.69 pF
744340300055	Power Inductors	WE-HCM / 55.00 nH / 52.90 A / DCR 0 mΩ / Cp 36.53 pF / Isat 23.60 A / SRF 75.0 MHz / MnZn	55 nH	0.27 mΩ	36.53 pF
744028000056	Power Inductors	WE-TPC / 56.00 nH / 4.50 A / DCR 12 mΩ / Cp 0.03 pF / Isat 6.00 A / SRF 2.0 GHz / NiZn	56 nH	11.5 mΩ	0.03 pF
744300006	Power Inductors	WE-HCM / 60.00 nH / 80.80 A / DCR 0 mΩ / Cp 26.65 pF / Isat 63.00 A / SRF 123.9 MHz / MnZn	60 nH	0.22 mΩ	26.65 pF
		WE-MAIA / 68.00 nH /			

Simulate NCD57090ADWR2G

Hold Ctrl to zoom

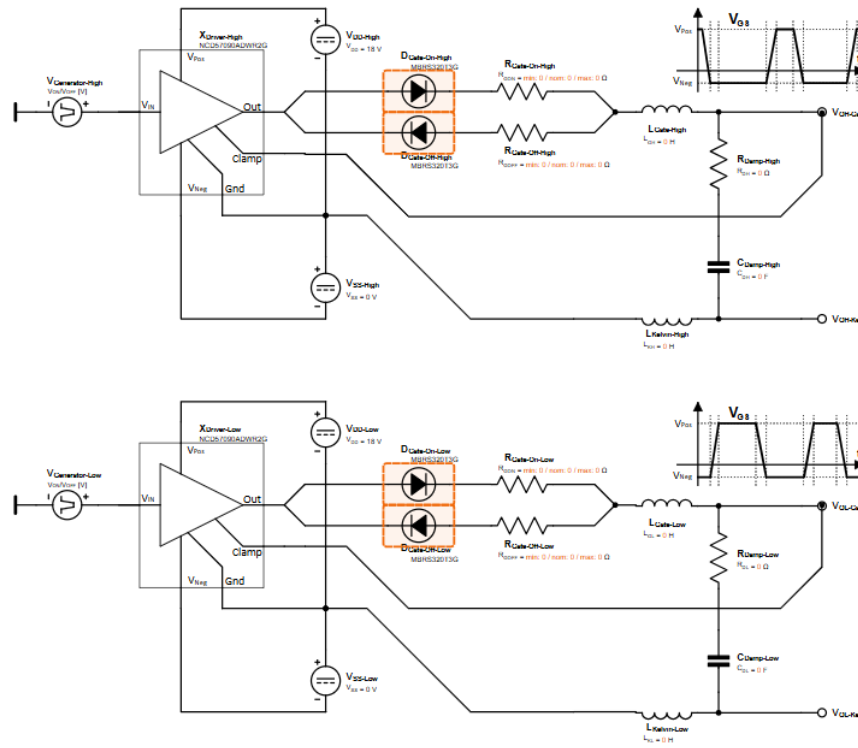
Simulate

After clicking the passive component, a selection window opens

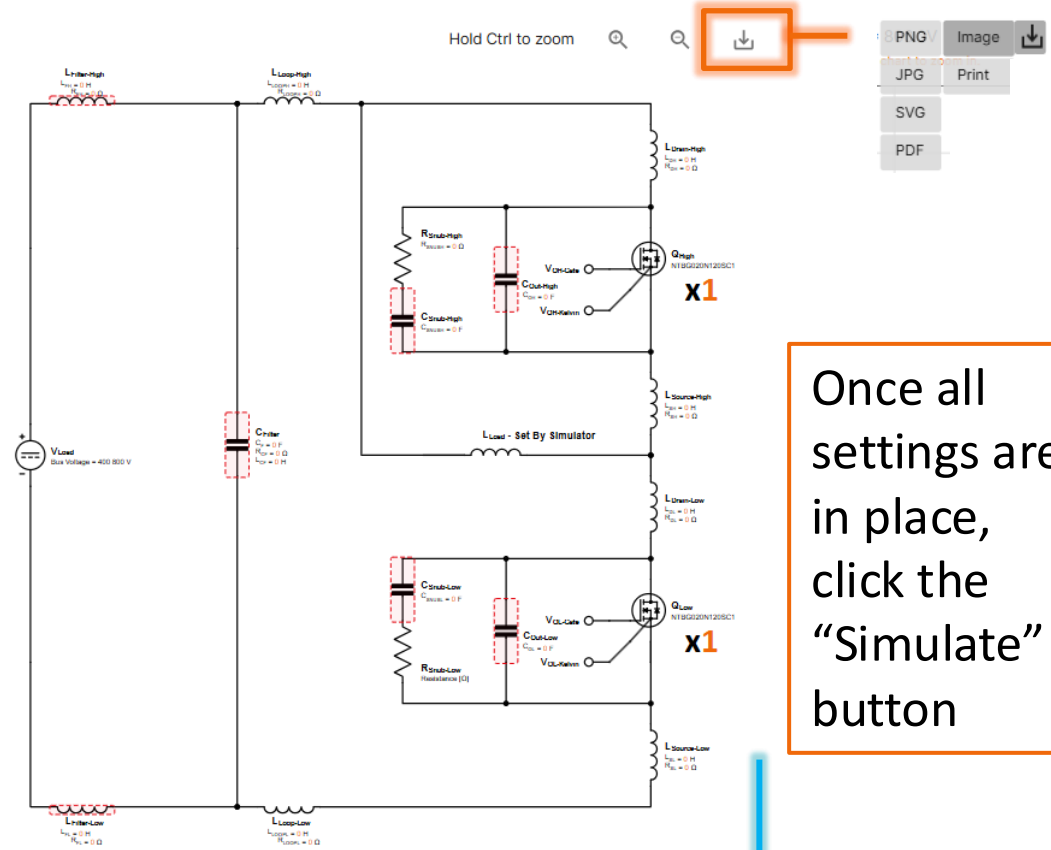
# Step 3: Running the SPICE simulations

Click the orange text in the schematic to edit parameters

Click the red dashed box around a component to select a Würth part



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



Once all settings are in place, click the "Simulate" button

Simulate NCD57090ADWR2G & NTB020N120SC1

# 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”

You can return at Stage 1 at any moment

The screenshot displays a complex circuit diagram with various components like resistors, capacitors, and inductors. A central dialog box titled "SSPMG Simulation" is open, containing the following text: "SSPMG Simulation with id: 149 has status PENDING! You can close this popup and monitor the simulation progress in the menu at the top left." Below this, there is an "Email notification (optional)" section with a text input field for "Notification email" and a checkbox labeled "Notify me by email when the simulation completes". At the bottom of the dialog, it says "You can also close this popup now — your simulation will continue running and results will appear in the menu at the top left." The background interface includes a "Simulate" button and a progress bar with three stages: "Choose", "Simulate", and "Analyze". The "Simulate" stage is currently active, and a "Back" button is highlighted in a red box.

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

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

The screenshot shows the onsemi simulation software interface. On the left, a 'Simulation Results' panel lists several 'SSPMG Simulation' entries with their status (Running, Completed, Failed) and timestamps. An orange box highlights this list. On the right, the main workspace displays the 'NTBG020N120SC1' MOSFET model, including its pinout diagram, a physical component image, and a circuit schematic. Below the model, there are configuration options for 'Eon/Eoff' (Double-Pulse Equivalent Loss Model), 'Conduction Diode Current (A)', 'Switching Current (A)', 'Power stage' (Transistors), 'Temperature List (°C)', and 'Bus Voltage List (V)'. An orange box highlights the 'Number of simulations still running' text above the interface.

e-mail received after the simulation is successfully complete

The email notification is from onsemi and states: 'Simulation complete'. It says: 'Your Elite Pairing Studio simulation has finished. Click the button below to open your results.' Below this is a 'View Results' button and a link: 'https://www.onsemi.com/design/tools-software/gate-driver-kit/NTBG020N120SC1?simulationId=149'. An orange box highlights this email notification.

# 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

**Simulation Results** Pair Parameters:

Simulation Name	Status	Time	Action
SSPMG Simulation	Running...	5/30/2026, 7:22:16 AM	
SSPMG Simulation	Completed	5/16/2026, 9:57:01 AM	Analyze
SSPMG Simulation	Completed	4/22/2026, 4:48:37 PM	Analyze
SSPMG Simulation	Completed	4/22/2026, 10:50:25 AM	Analyze
SSPMG Simulation	Failed	5/23/2026, 1:31:25 PM	Error
SSPMG Simulation	Completed	5/16/2026, 9:57:01 AM	Analyze
SSPMG Simulation	Completed	4/22/2026, 4:48:37 PM	Analyze
SSPMG Simulation	Completed	4/22/2026, 10:50:25 AM	Analyze
SSPMG Simulation	Completed	4/21/2026, 9:19:13 AM	Analyze
SSPMG Simulation	Completed	4/7/2026, 10:41:27 AM	Analyze

**NTBG020N120SC1**  
Silicon Carbide (SiC) MOSFET - EliteSiC, 20 mohm, 1200 V, M1, D2PAK-7L

[Datasheet](#)  
[Case Outline Drawing](#)

Conduction Diode Current (A):  
Min: 1 (A), Max: 50 (A), Step: 2 (A)

Switching Current (A):  
Min: 1 (A), Max: 50 (A), Step: 4 (A)

Power stage: **Transistors** | Diode & Transistor

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

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

# Outline of User Guide

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

# General View

Simulation Results Select & Analyze for NTB020N120SC1 & NCD57090ADWR2G

Simulation Results Selector:

Switching Current (A)

Bus Voltage (V)

Filter for range of switching current and bus voltage

Plots and parametric analysis of simulated waveforms

Transient Results (800.0 V, 50.0 A):

Cursor Measurements for NTB020N120SC1 & NCD57090ADWR2G

EMI and Overshoots

PLECS Model Plots:

At	$\Delta V_{DS(on)}$	$\Delta V_{DS(on)}/M$
0.0001489	1001	6.631e-6

At	$\Delta I_{DS(on)}$	$\Delta I_{DS(on)}/M$
0.0001489	1270	8.487e-6

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

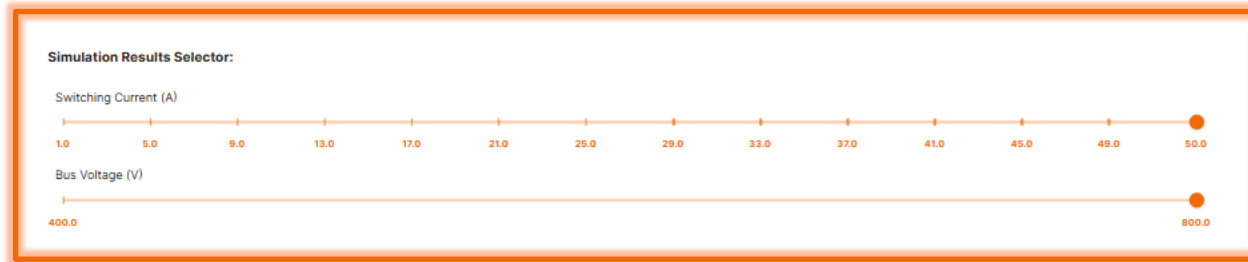
Results & Model: [Get PLECS Model](#)

PLECS sim: [Launch Elite Power Simulator](#)

Back Choose Simulate Analyze New Simulation

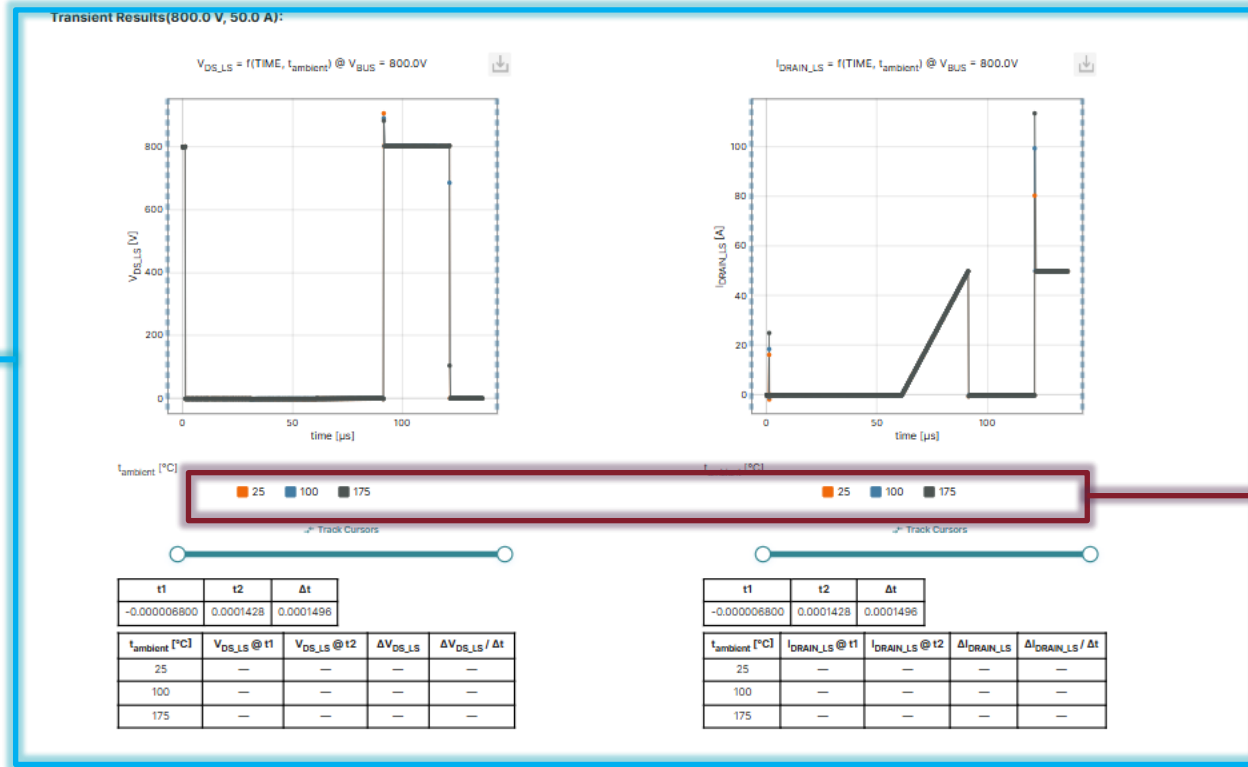
Links to PLECS model generator and Elite Power Simulator

# 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



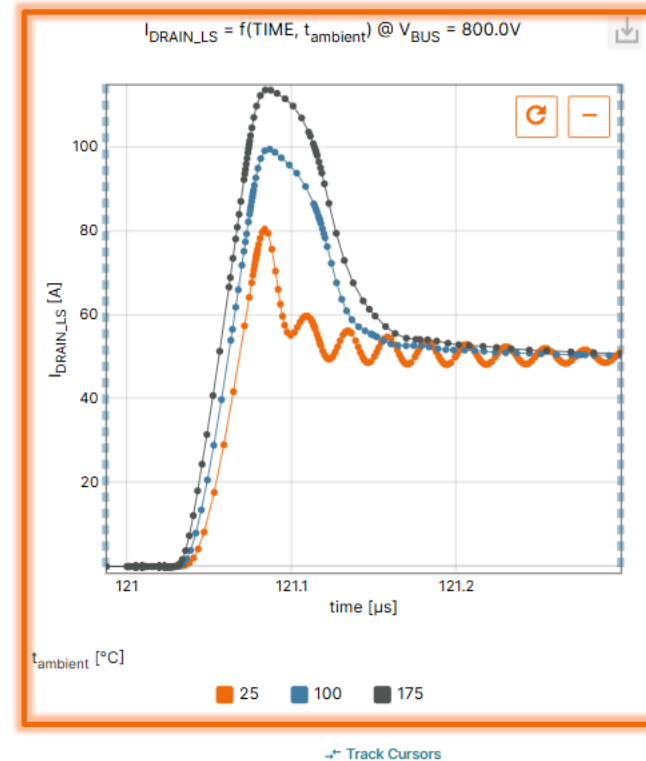
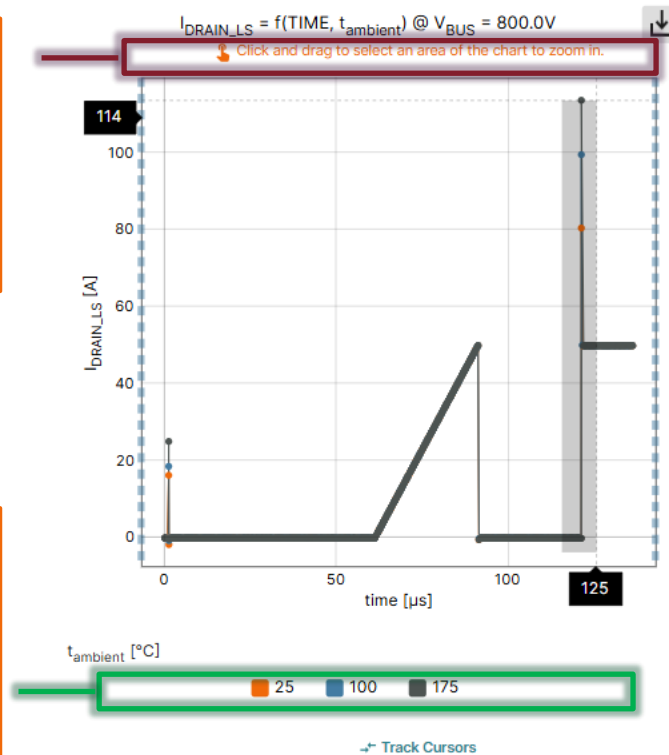
By scrolling down, 5 plots appear representing  $V_{DS\_LS}$ ,  $I_{\text{DRAIN\_LS}}$ ,  $V_{GS\_LS}$ ,  $V_{DS\_HS}$  and  $I_{\text{DRAIN\_HS}}$

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



Zoomed plot

t1	t2	Δt
-0.000006800	0.0001428	0.0001496

$t_{ambient} [^{\circ}C]$	$I_{DRAIN\_LS} @ t1$	$I_{DRAIN\_LS} @ t2$	$\Delta I_{DRAIN\_LS}$	$\Delta I_{DRAIN\_LS} / \Delta t$
25	—	—	—	—
100	—	—	—	—
175	—	—	—	—

t1	t2	Δt
0.0001210	0.0001213	3.126e-7

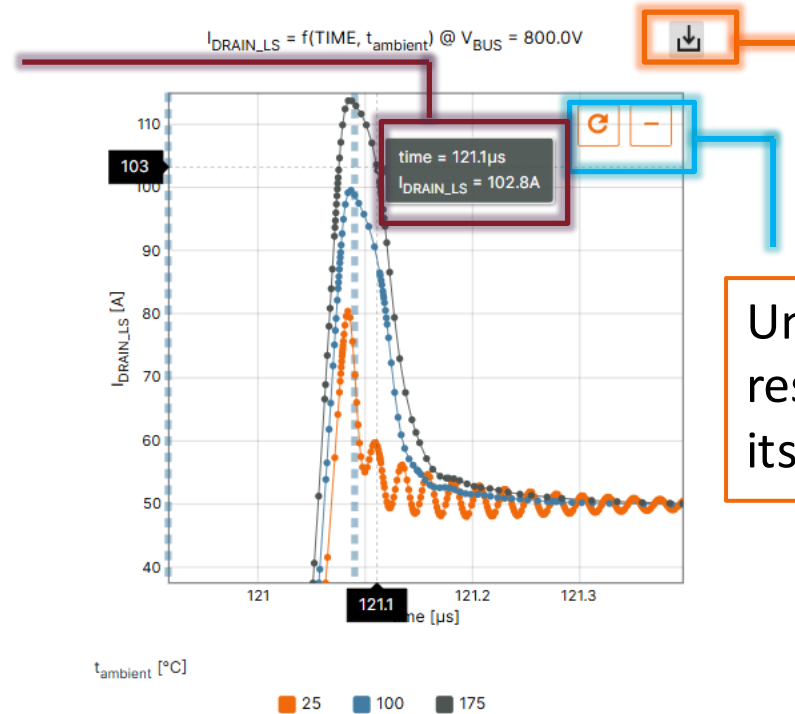
  

$t_{ambient} [^{\circ}C]$	$I_{DRAIN\_LS} @ t1$	$I_{DRAIN\_LS} @ t2$	$\Delta I_{DRAIN\_LS}$	$\Delta I_{DRAIN\_LS} / \Delta t$
25	0.000003190	51.20	51.20	1.638e+8
100	0.000002327	50.34	50.34	1.611e+8
175	0.000008363	50.82	50.82	1.626e+8

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

Track Cursors

t1	t2	Δt
0.0001209	0.0001211	1.744e-7

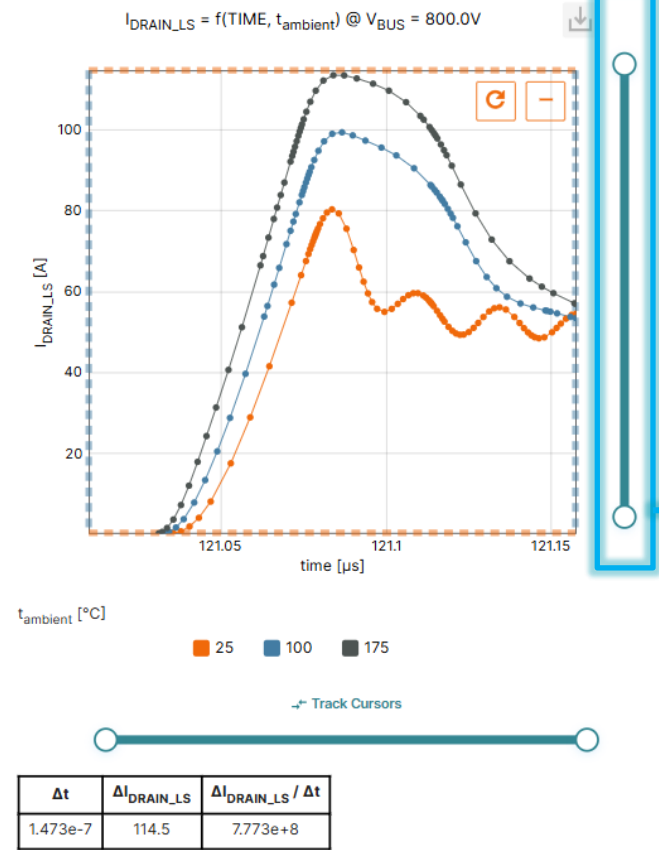
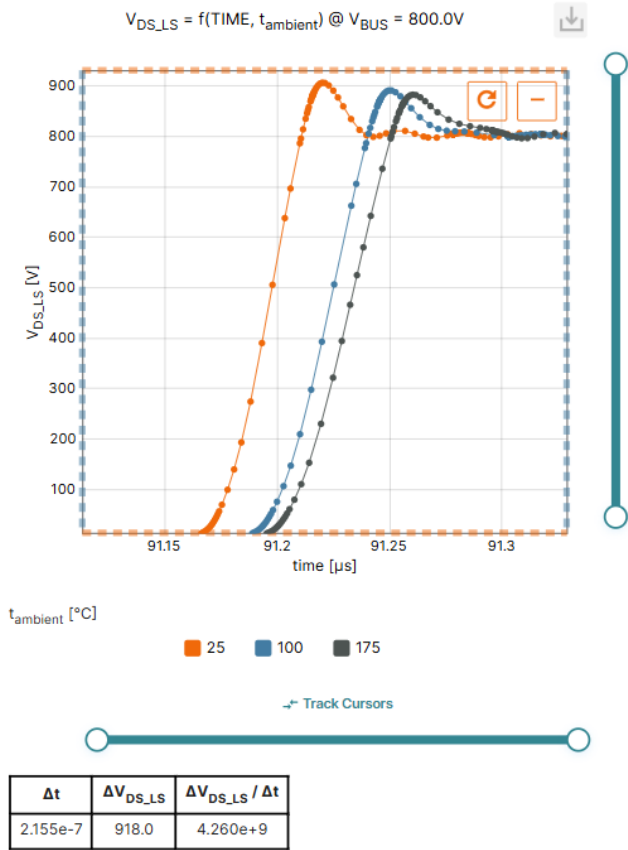
t <sub>ambient</sub> [°C]	I <sub>DRAIN_LS</sub> @ t1	I <sub>DRAIN_LS</sub> @ t2	ΔI <sub>DRAIN_LS</sub>	ΔI <sub>DRAIN_LS</sub> / Δt
25	—	68.62	—	—
100	—	98.53	—	—
175	—	113.0	—	—

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

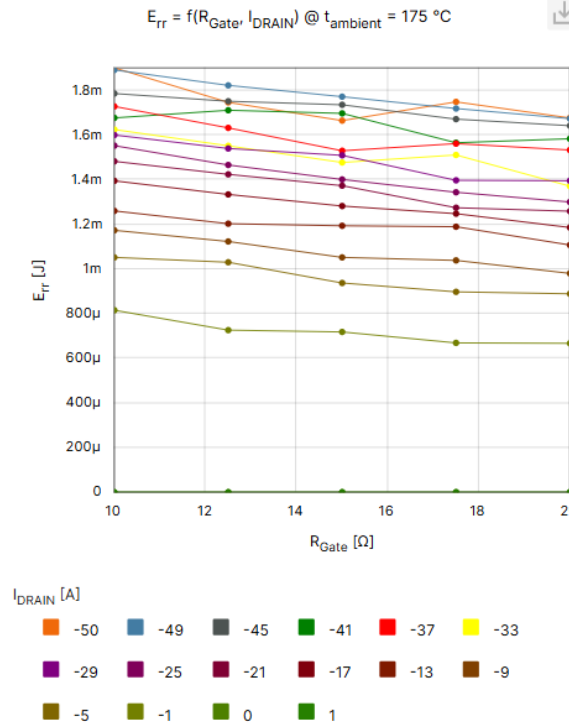
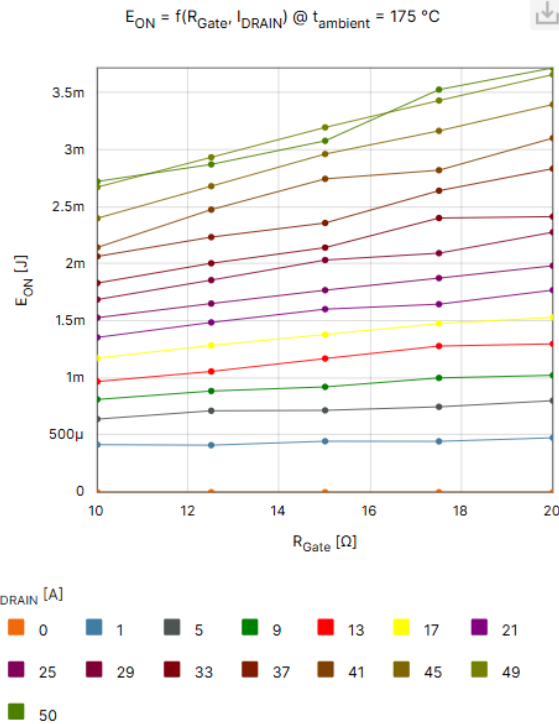
EMI and Overshoots



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

# Step 4: Interacting with the PLECS model plots

PLECS Model Plots:



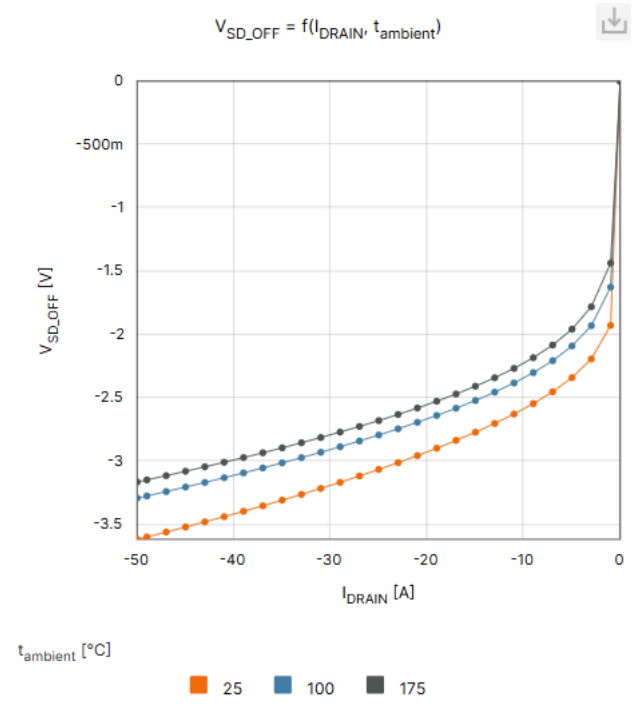
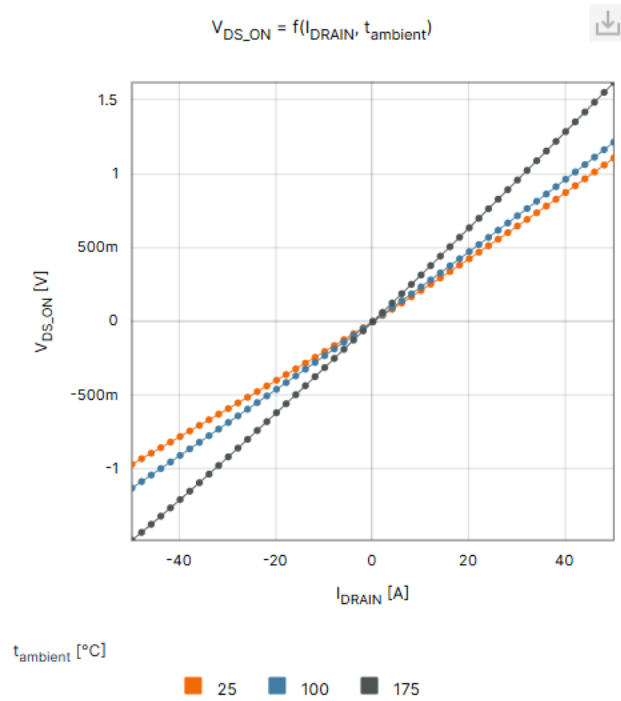
The interaction is similar as in the transient and overshoot plots

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)

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](http://www.onsemi.com/design/elite-pairing-studio)

# onsemi<sup>TM</sup>

Intelligent Technology. Better Future.

Follow us @onsemi



[onsemi.com](https://www.onsemi.com)