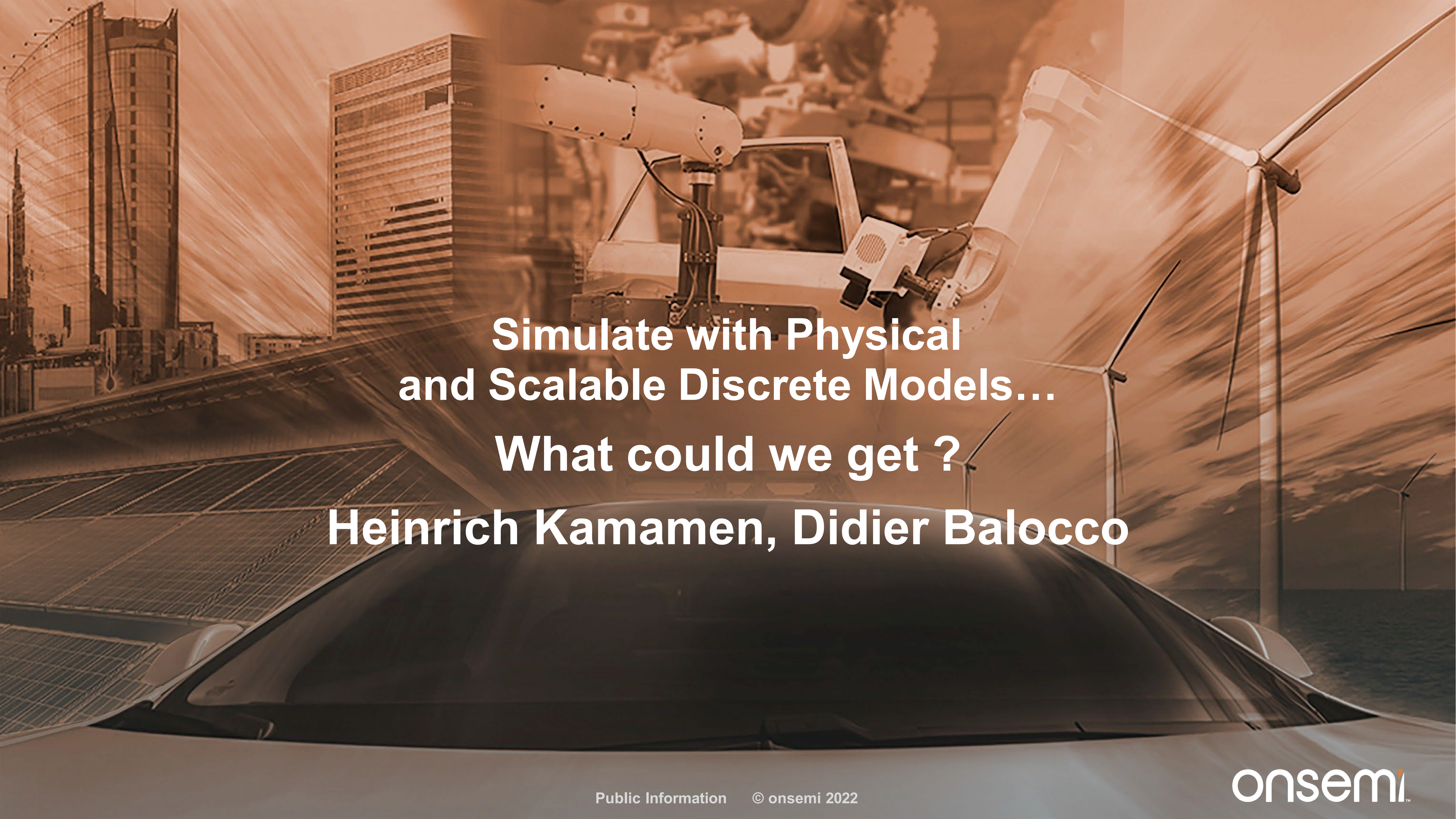


Simulate with Physical and Scalable Discrete Models…

What could we get ?

Heinrich Kamamen, Didier Balocco

Content

- ○ Simulate Devices to Extract Parameters
  - ▶ On-Region simulation
  - ▶ $R_{DS(ON)}$ simulation
  - ▶ Transfer characteristic
  - ▶ Output Capacitor
    - ■ Small Signal, Effective, Energy related or Charge related
  - ▶ Breakdown Voltage (and Drain Leakage Current)
- ○ Application Simulations
    - ■ Evaluate Losses and Junction Temperature on a DC-DC Boost

Simulate devices : Let us practice !

? Is it easy and useful ?

? What can kind of results could be obtained ?

On-Region simulation

Drain Current vs Drain-to-Source Voltage

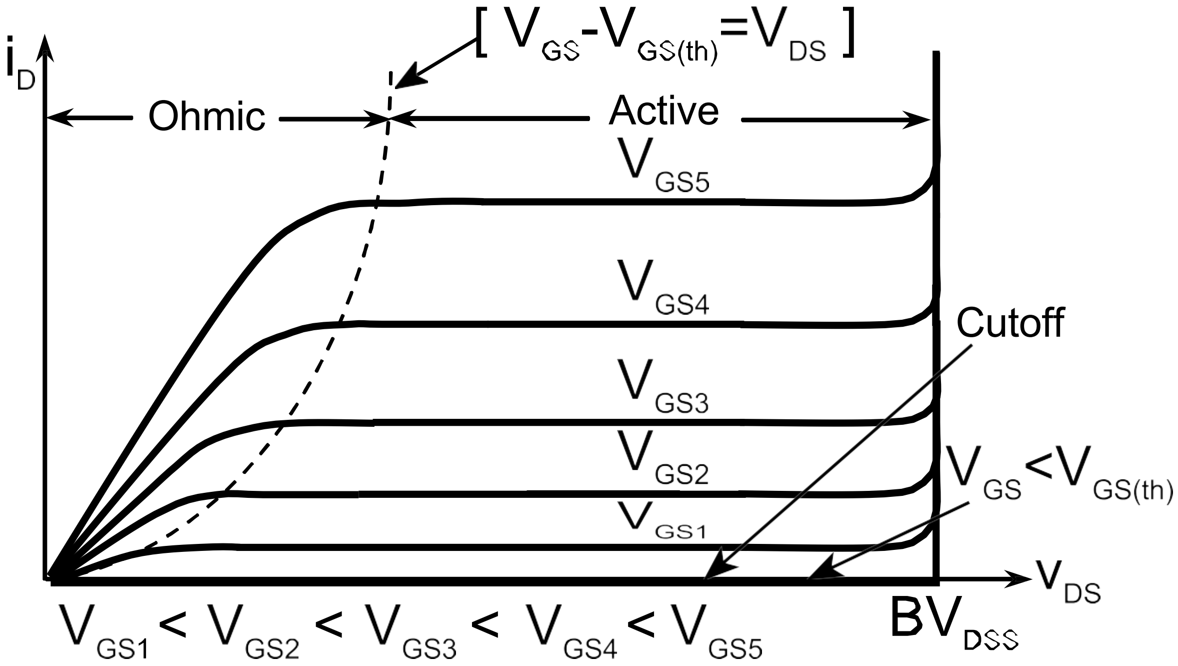
MOSFET On-Region – Books' Curve

In books, we found the following on-region curves.

We have two regions:

Ohmic and Active

This is well known but in practice, this curve can be slightly different.

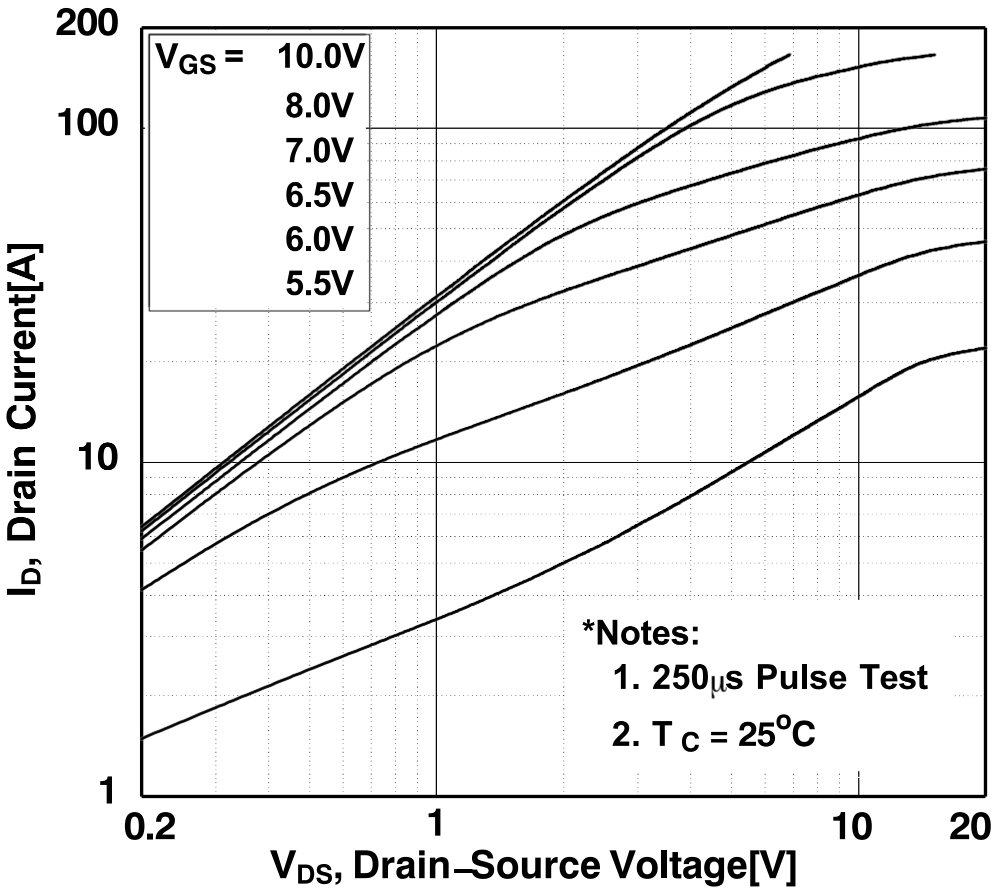


MOSFET On-Region – Data Sheet Curve

For the NTHL040N65S3F, SuperFET3 Fast recovery 40 mν

The data sheet give the following curve with Log-Log scales.

This curve is done with a 250-μs pulse test to avoid the MOSFET to heat…



*Notes:
1. 250μs Pulse Test
2. $T_C = 25^{o}C$

MOSFET On-Region - Schematic

The schematic is very simple :

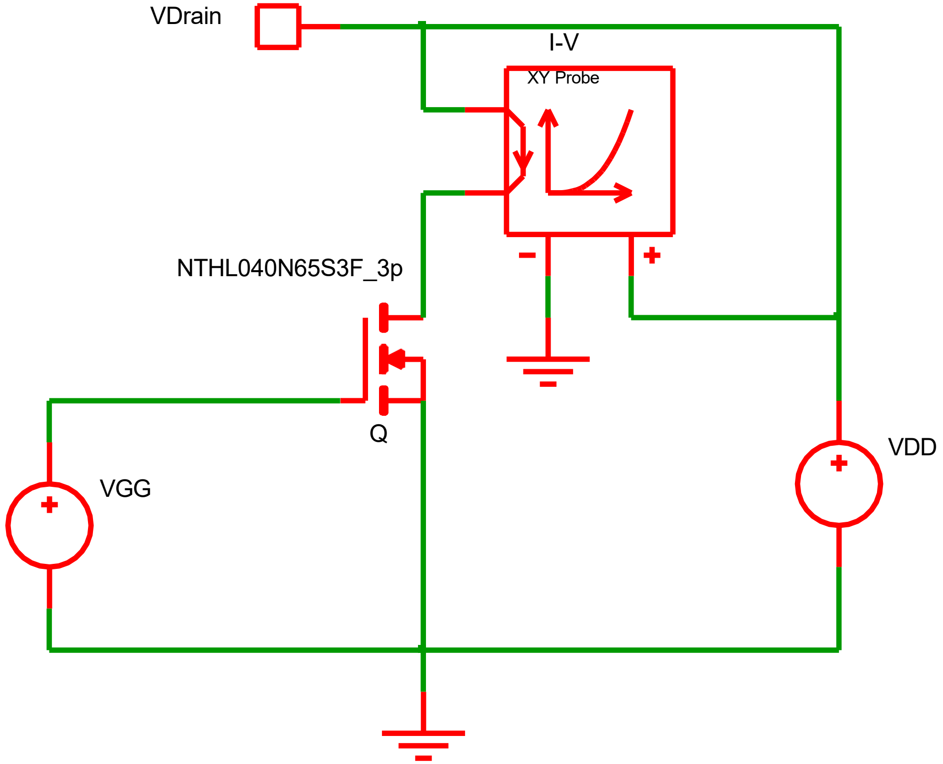
The MOSFET

A Drain-to-Source voltage source

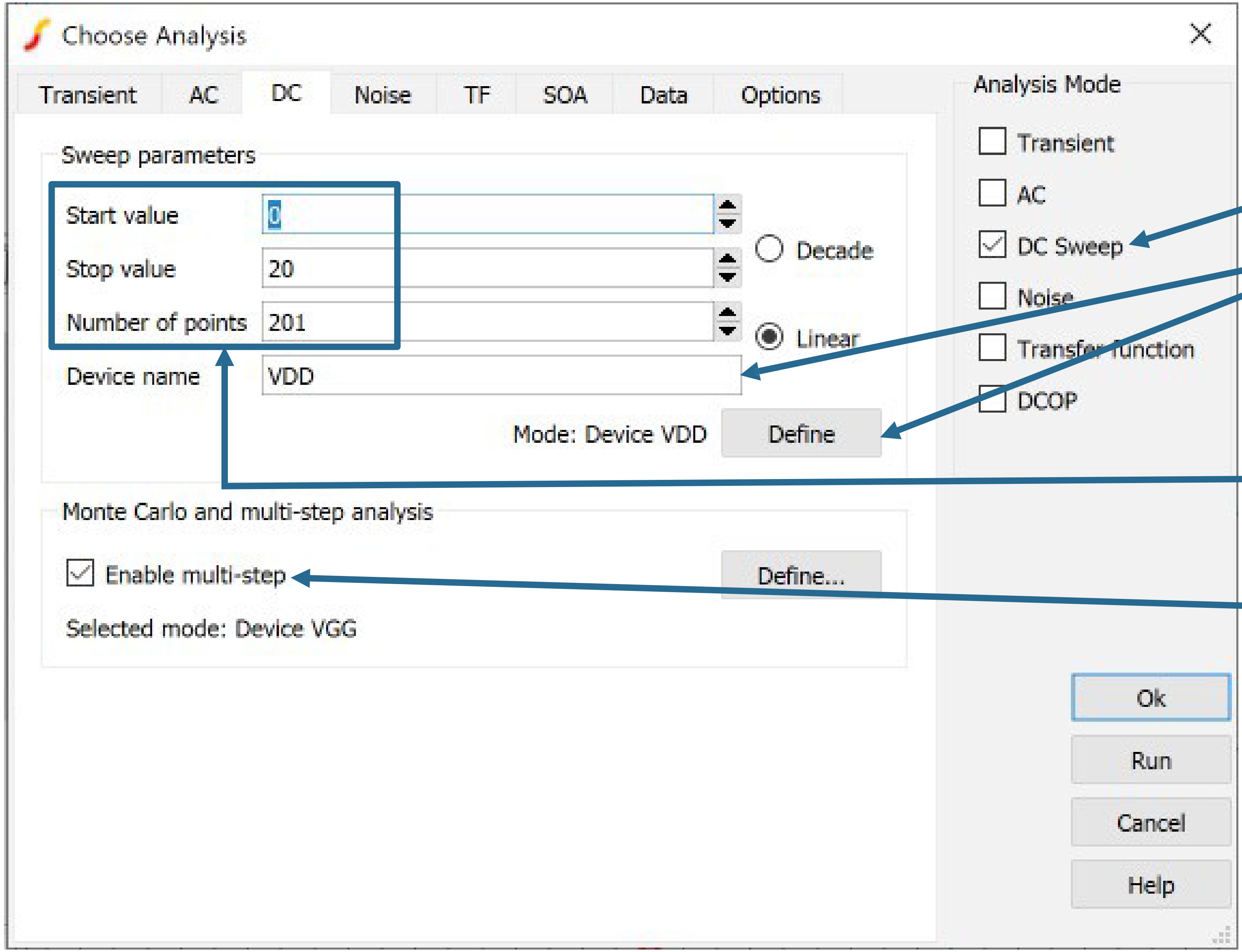
A Gate-to-Source voltage source

That’s all you need…

We add a “X-Y Probe” pseudo-component to pre-set the output graph.



MOSFET On-Region – Setup 1



Choose Analysis

Sweep parameters

Start value: 0

Stop value: 20

Number of points: 201

Device name: VDD

○ Decade

● Linear

Mode: Device VDD

Monte Carlo and multi-step analysis

☑ Enable multi-step

Selected mode: Device VGG

Analysis Mode

☐ Transient

☐ AC

☑ DC Sweep

☐ Noise

☐ Transfer function

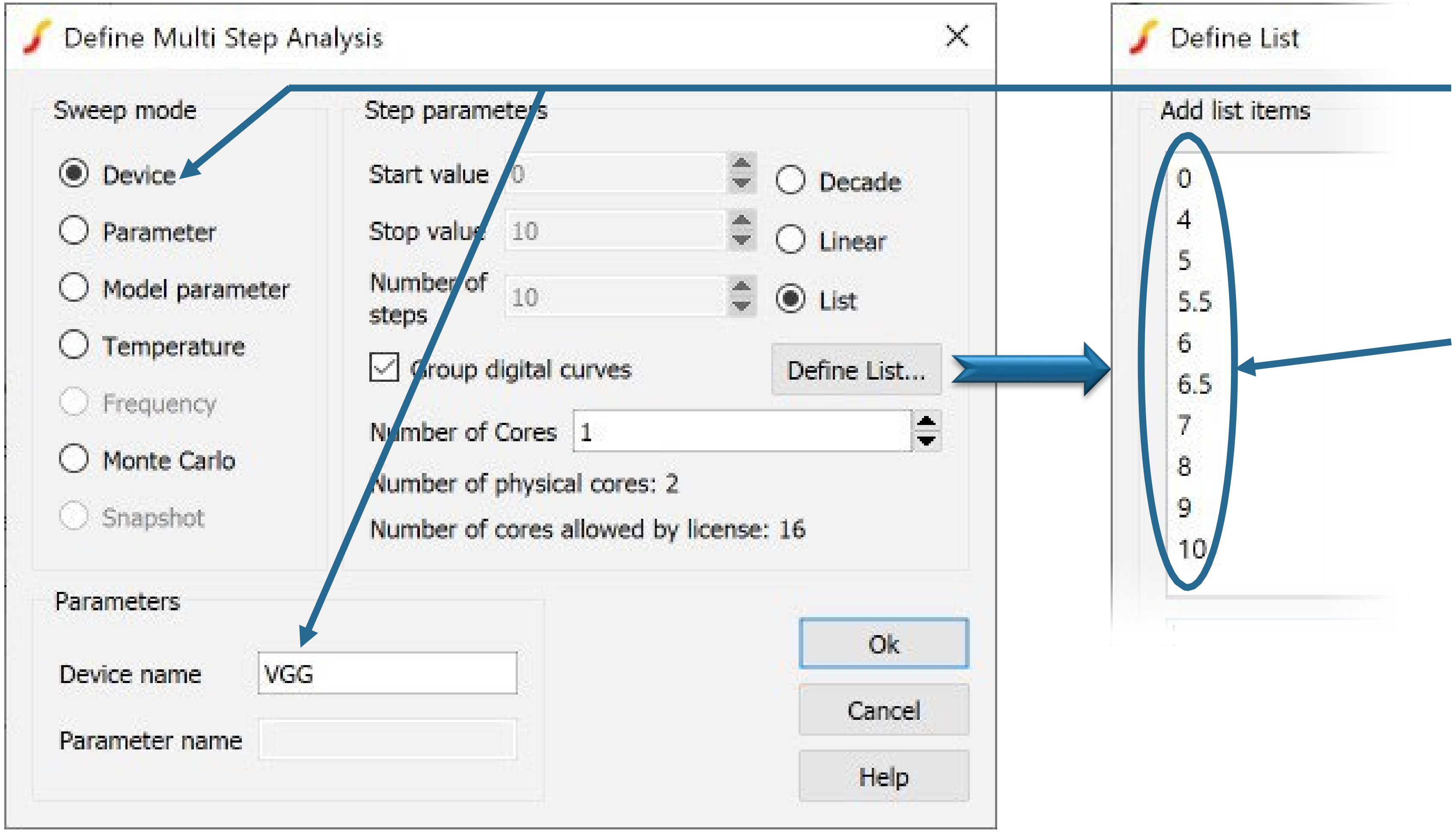
☐ DCOP

We define the simulation as "DC Sweep" on "VDD" source

We define the variation amplitude.

We enable the "Multi-Step" analysis to vary the "VGG" source by discrete steps

MOSFET On-Region – Setup 2



Start value: 0

Stop value: 10

Number of steps: 10

○ Decade
○ Linear
● List

☑ Group digital curves

Number of Cores: 1

Number of physical cores: 2

Number of cores allowed by license: 16

Parameters

Device name: VGG

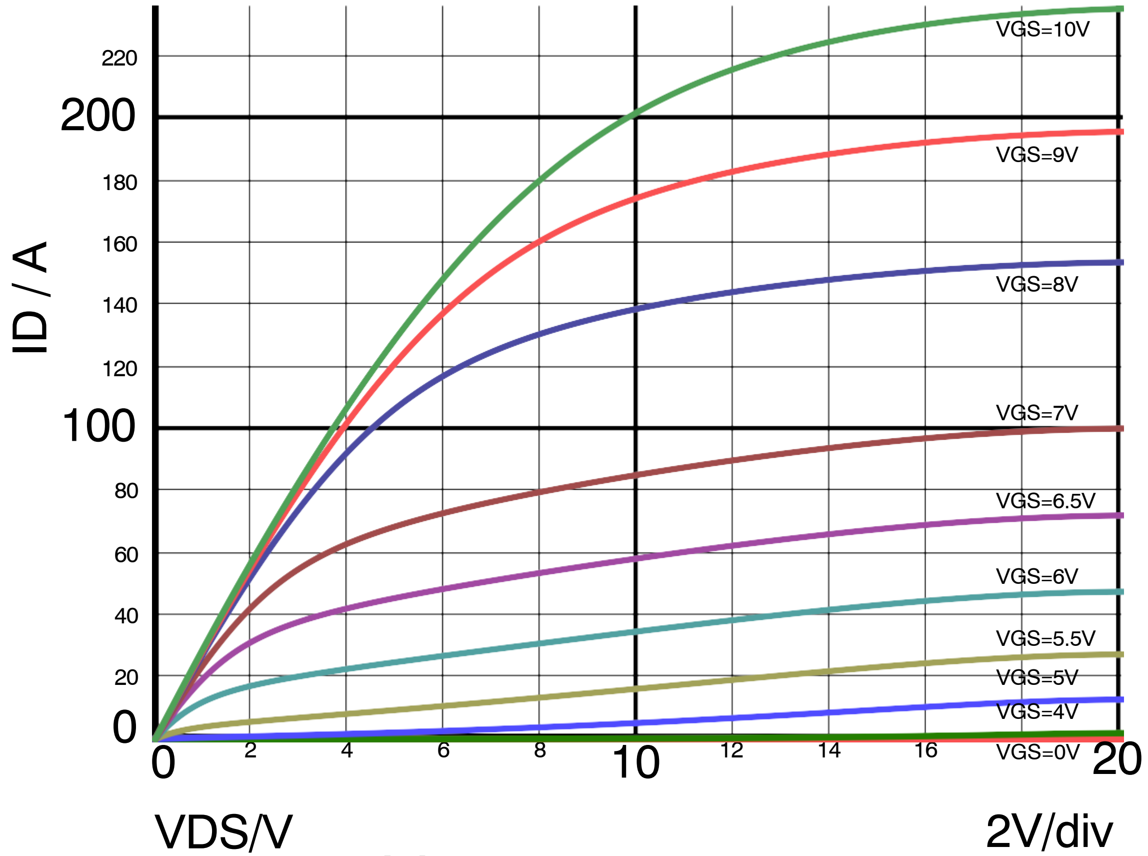
Parameter name:

0
4
5
5.5
6
6.5
7
8
9
10

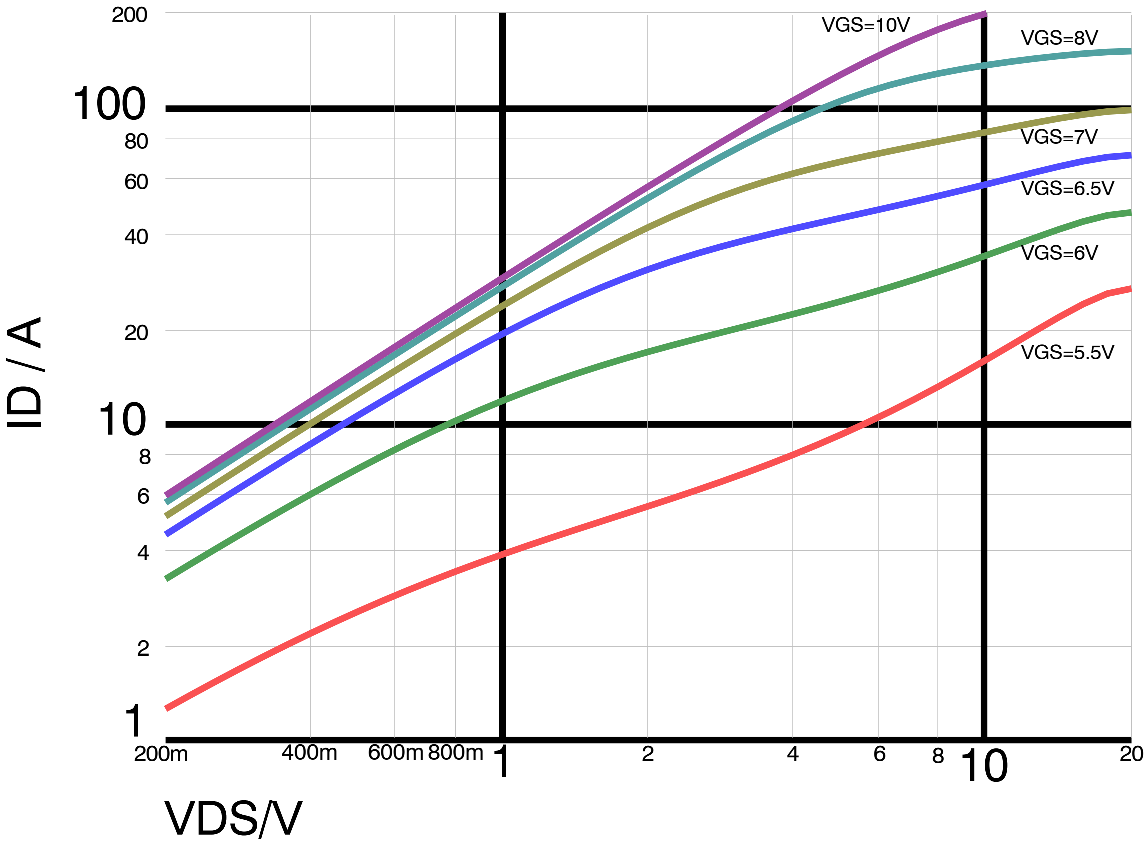
We define the values "VGG" voltage source can take as a list.

We have the same list as the specification.

MOSFET On-Region - Results



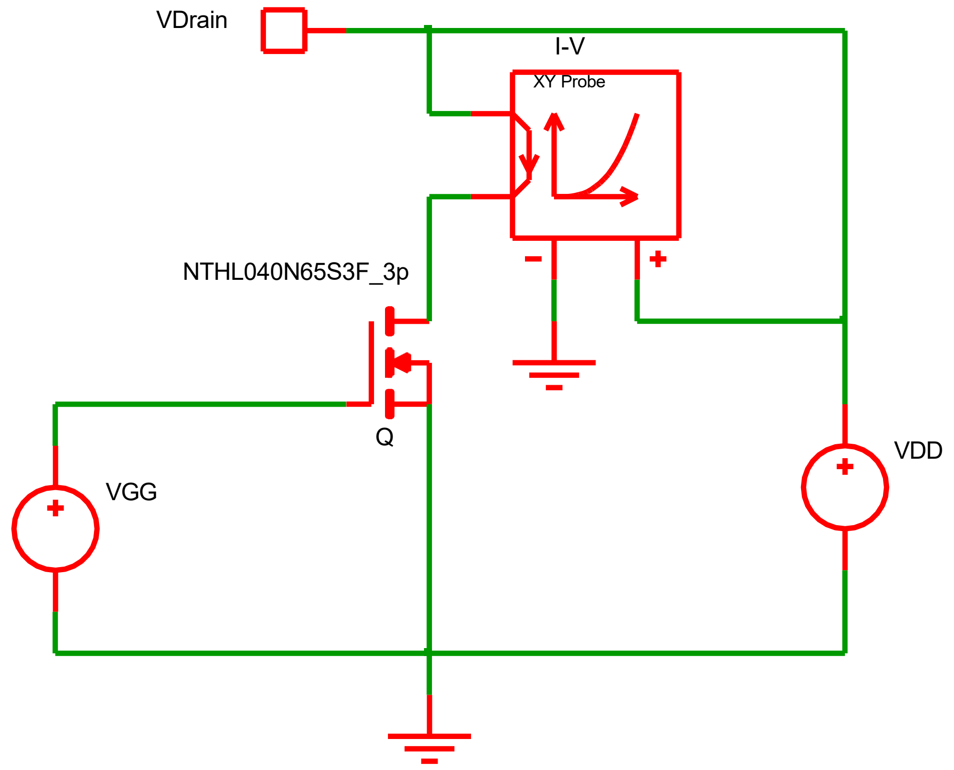
Linear scales



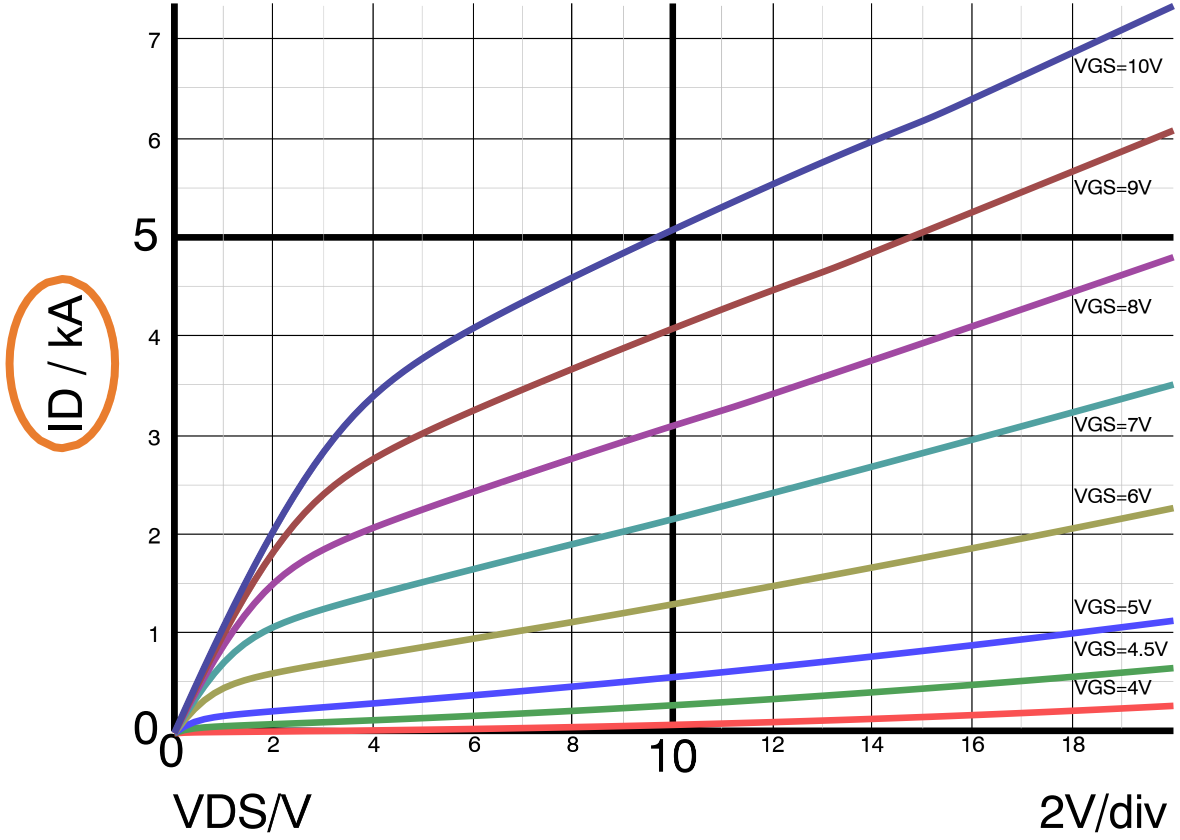
Logarithmic scales

MOSFET On-Region - Schematic

We re-use the same simulation schematic for a Medium Voltage MOSFET : NTMFS5C604N.



MOSFET On-Region - Results



The results are far above maximum current capability given in the specification for the MOSFET

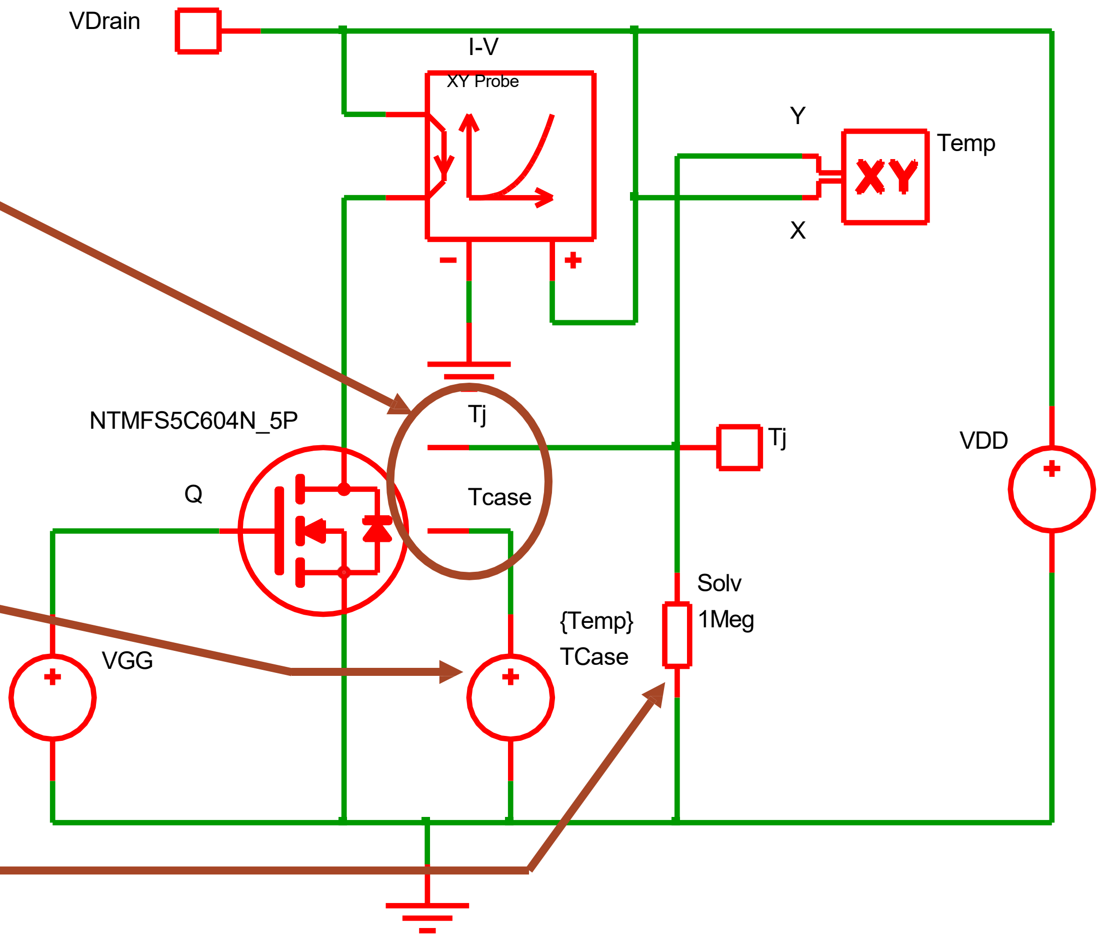
The active region is not really horizontal

MOSFET On-Region with 5-pin model - Schematic

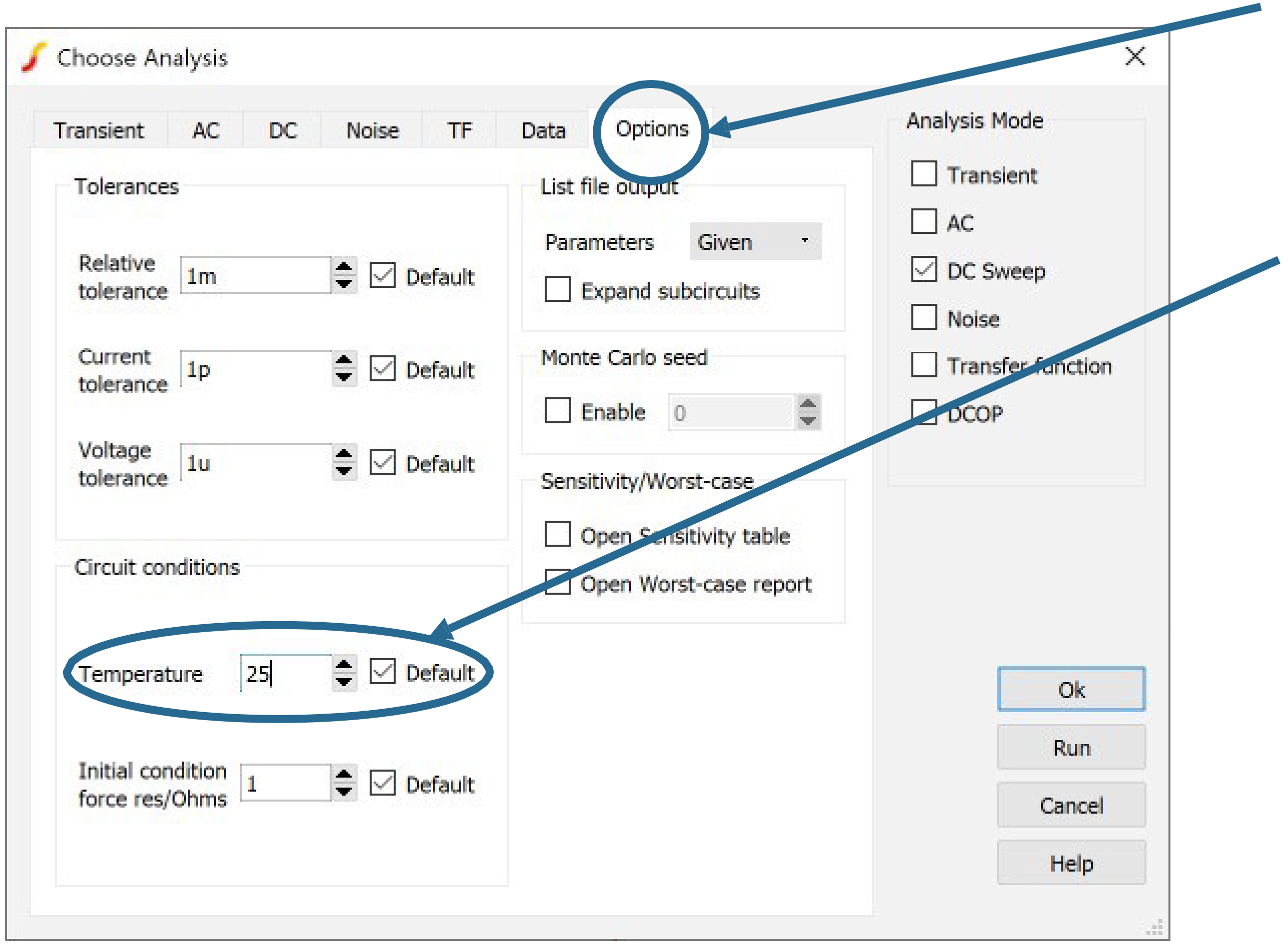
We will use a 5-pin device to see the temperature behavior during this test.

The model uses the electro-thermal equivalence where Voltage represents the Temperature and the Current represents the Power dissipated. The case temperature is set to the system temperature using a voltage source. The voltage-source value is set with the SIMetrix variable “Temp” storing the system temperature.

We need to add a 1-Mν resistor on the “Junction Temperature” pin to help the solver to converge.



MOSFET On-Region with 5-pin model – Temperature Setup



Choose Analysis

Tolerances

Relative tolerance: 1m ☑ Default

Current tolerance: 1p ☑ Default

Voltage tolerance: 1u ☑ Default

Circuit conditions

Temperature: 25 ☑ Default

Initial condition force res/Ohms: 1 ☑ Default

List file output

Parameters: Given

☐ Expand subcircuits

Monte Carlo seed

☐ Enable 0

Sensitivity/Worst-case

☐ Open Sensitivity table

☐ Open Worst-case report

Analysis Mode

☐ Transient

☐ AC

☑ DC Sweep

☐ Noise

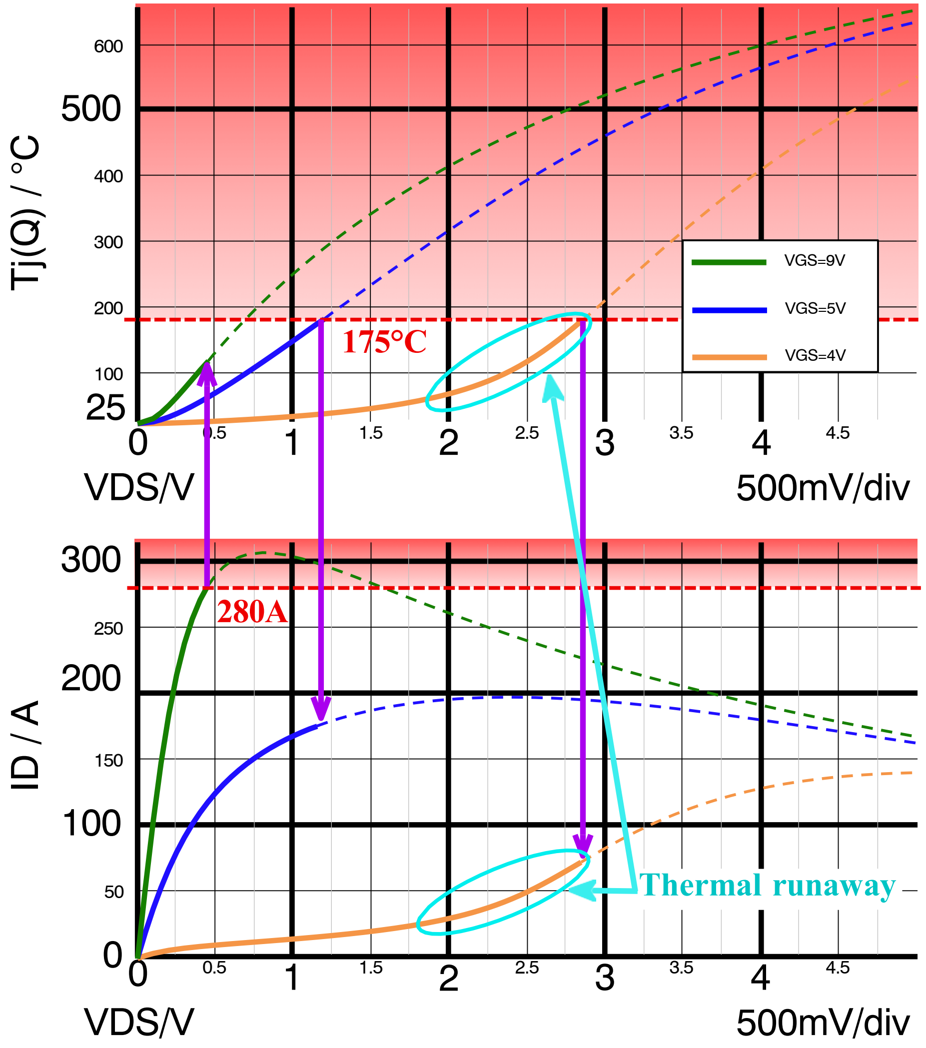
☐ Transfer function

☐ DCOP

In the option tab inside simulation setup windows,

The system temperature can be set to 25 °C

MOSFET On-Region with 5-pin model - Results



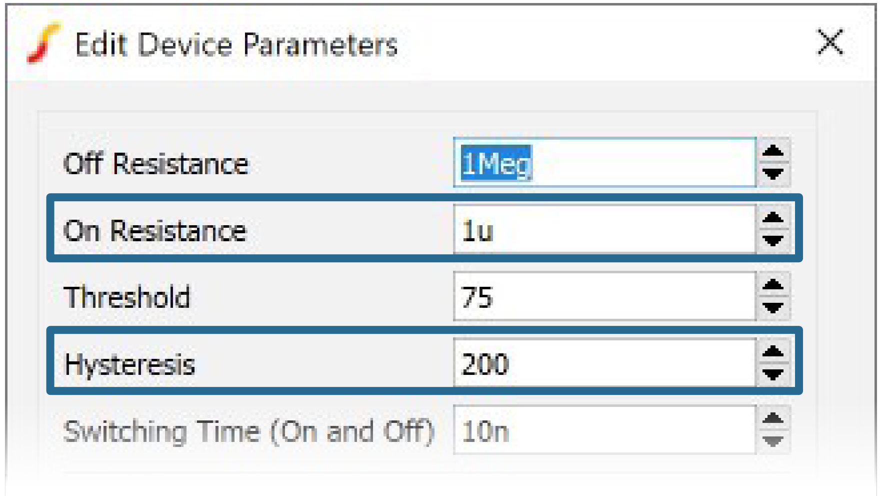
For V_{GS} = 4 V, we see a thermal runaway in the active region. The junction temperature rises above 175°C before the drain current reaches the 280 A specification limit.

For V_{GS} = 5 V, the device stays in the ohmic region, but the junction temperature rises to 175°C before it gets to 280-A drain current limit.

For V_{GS} = 9 V, the device stays in the ohmic region, but the drain current rises to 280 A limit before it gets to 175°C junction temperature.

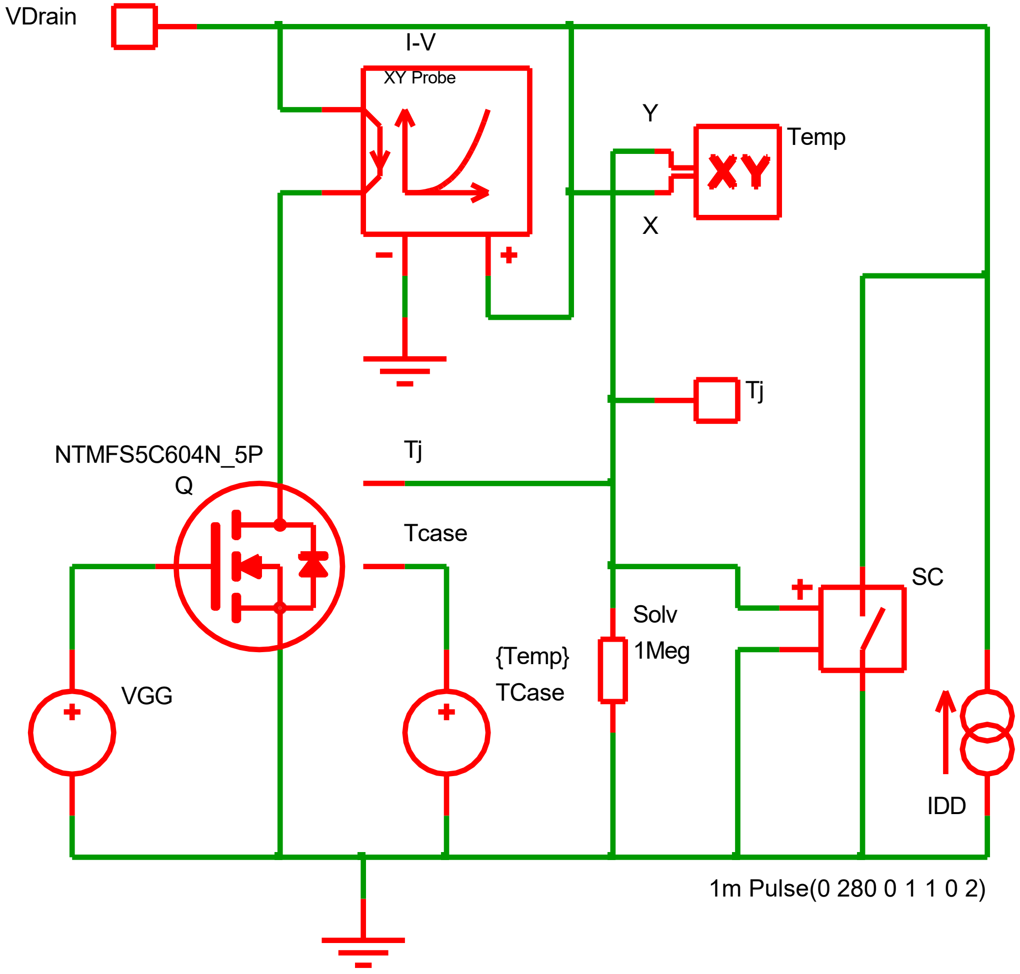
MOSFET On-Region within the limits - Schematic

- We set IDD current source to 280 A.
- We use a switch with very low on-resistance driven by the junction temperature to short the current source when the temperature gets above 175°C.
- We implement a very large hysteresis to avoid a new turn on when the device cools down.

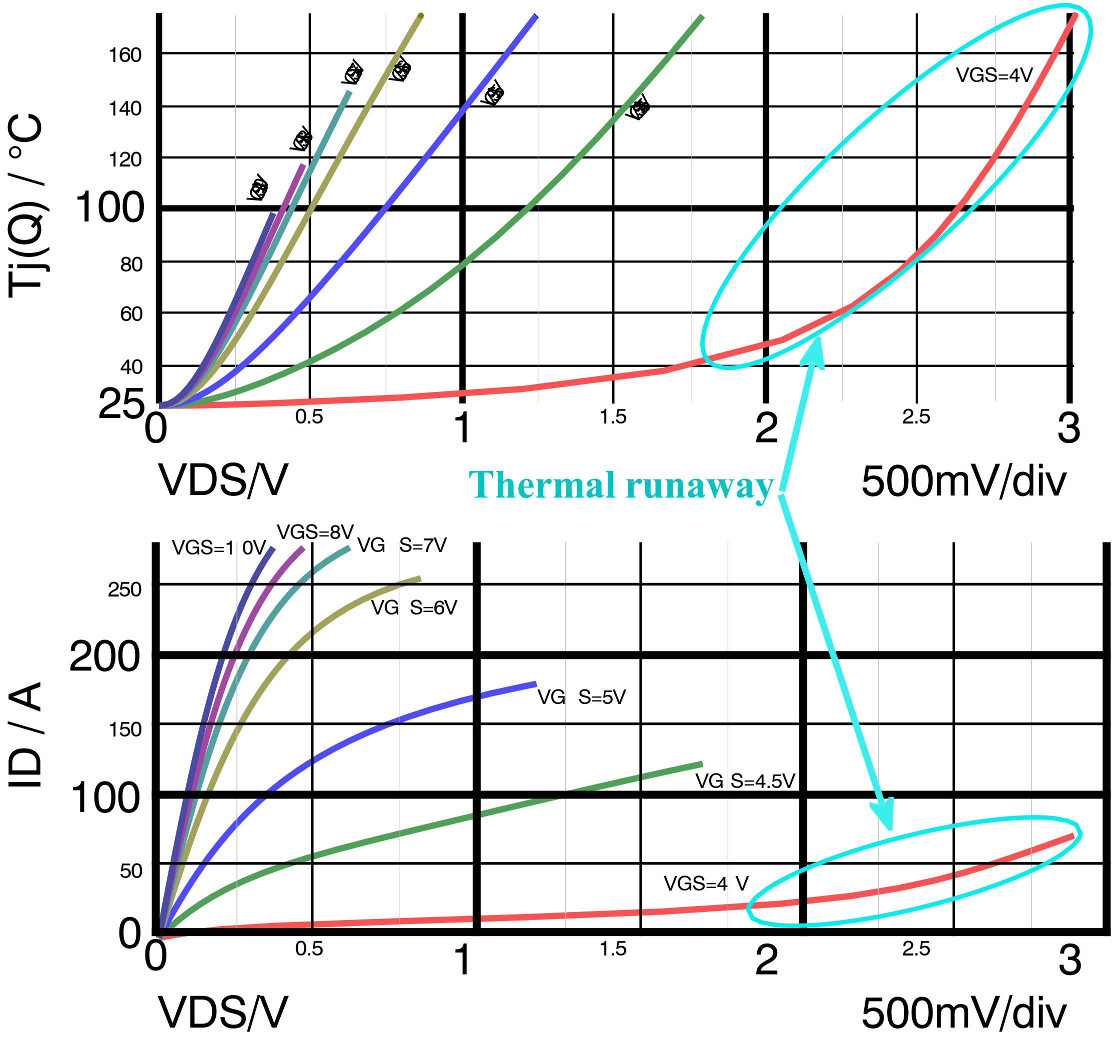


Edit Device Parameters

Off Resistance	1Meg
On Resistance	1u
Threshold	75
Hysteresis	200
Switching Time (On and Off)	10n



MOSFET On-Region within the limits - Results



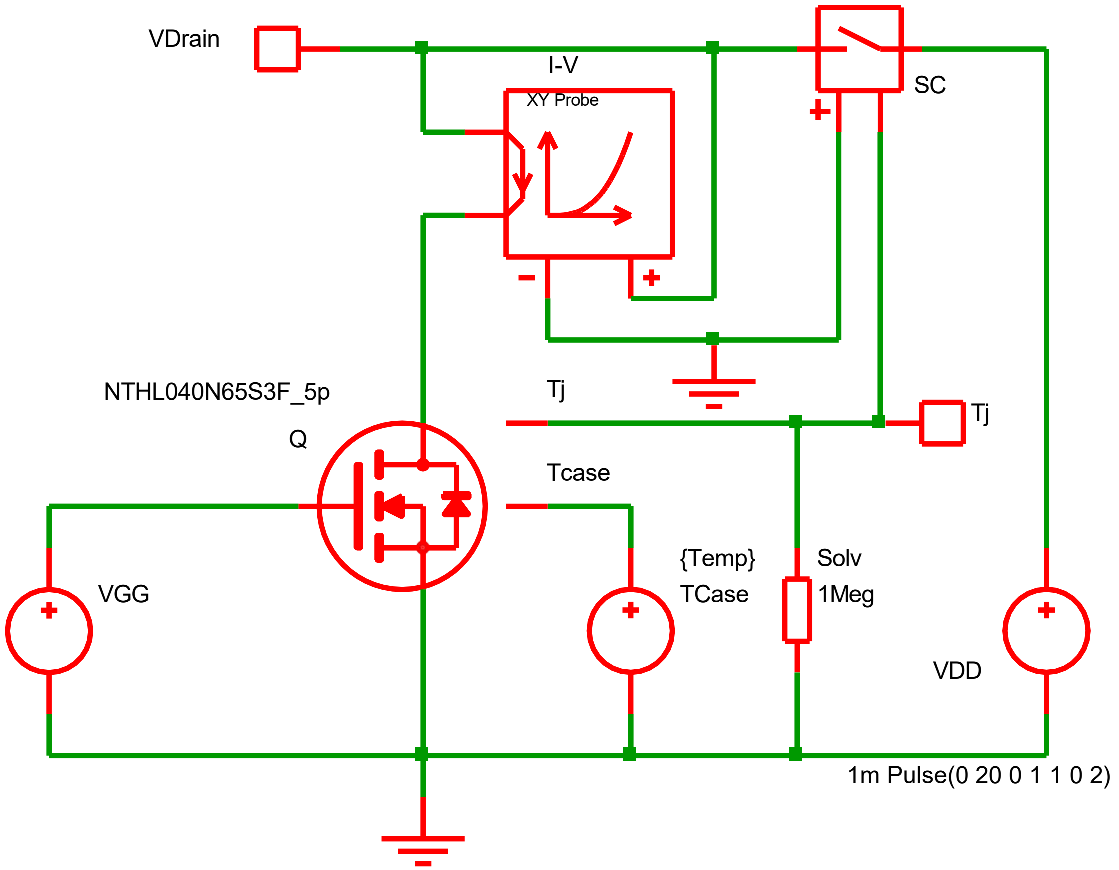
- This is the real useful curves for 25°C package temperature.
- Using simulation, we can have the real operating curves in real conditions.

MOSFET On-Region within the limits - Schematic

With the NTHL040N65S3F, we can applied the same setup.

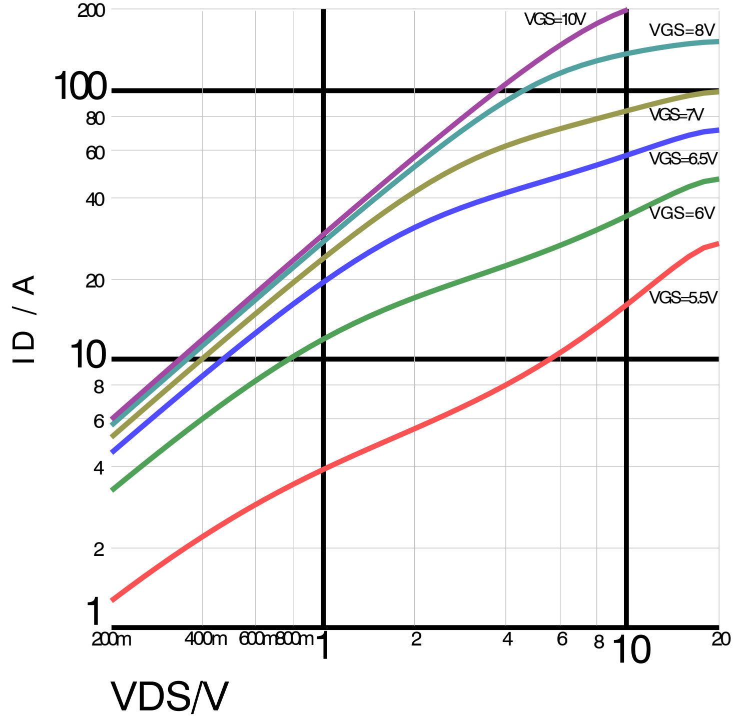
We use a switch with very high off-resistance driven by the junction temperature to open the voltage source when the temperature gets above 150°C.

We implement a very large hysteresis to avoid a new turn on when the device cools down.

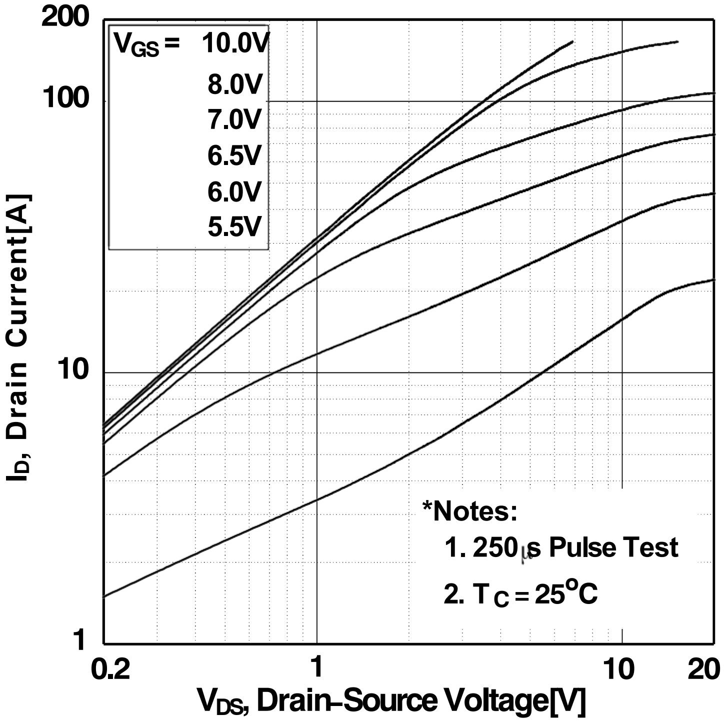


Comparing I_D-V_{DS} graph for the NTHL040N65S3F

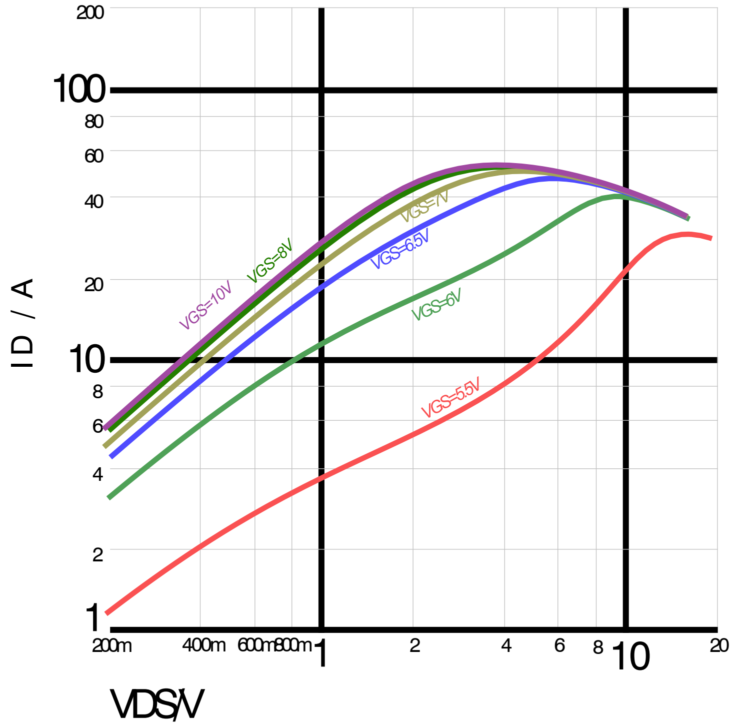
3 pins model



Data Sheet



5 pins model



- 3 different results…
  - With 3 pins model, the die is "some how" maintained at constant temperature (or like if measurement was done with an infinite small pulse)
  - Data sheet use 250µs pulse for measurement.
  - With 5 pins model, measurement are done in DC (infinite long pulse)

$R_{DS(ON)}$ simulation

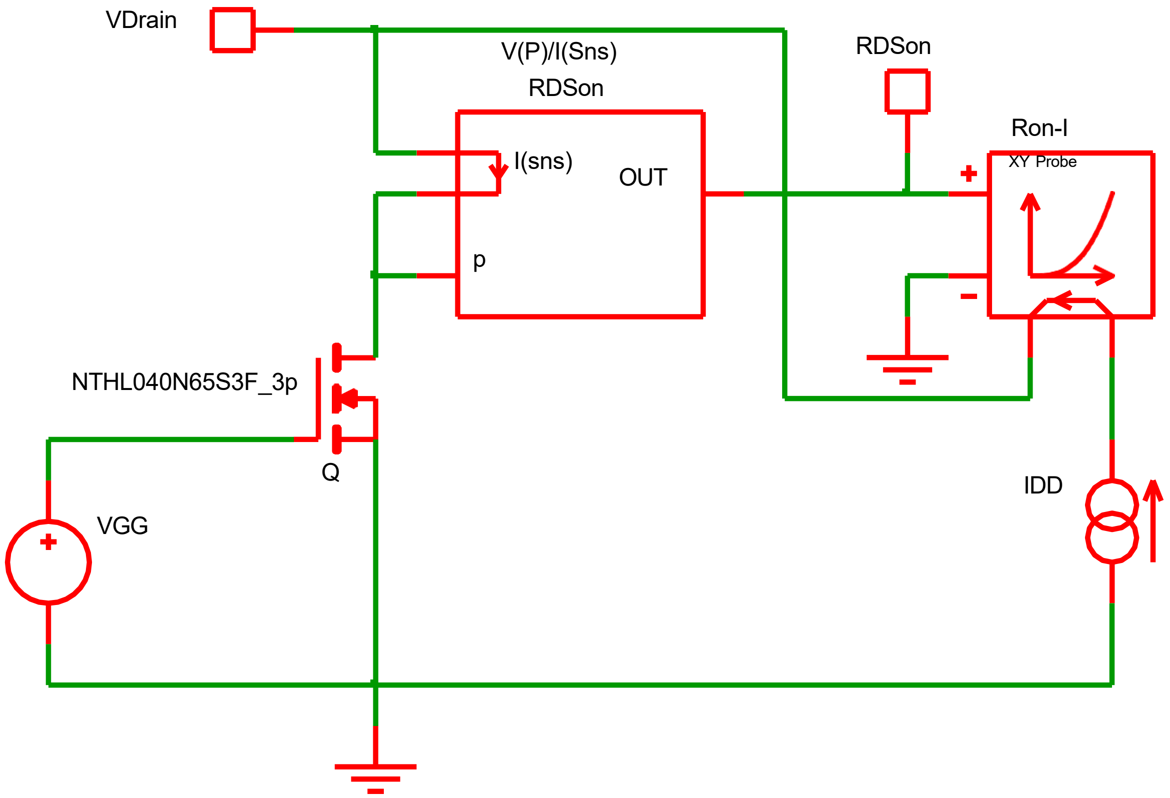
vs Drain Current or Temperature or Time

$R_{DS(ON)}$ vs Drain Current - Schematic

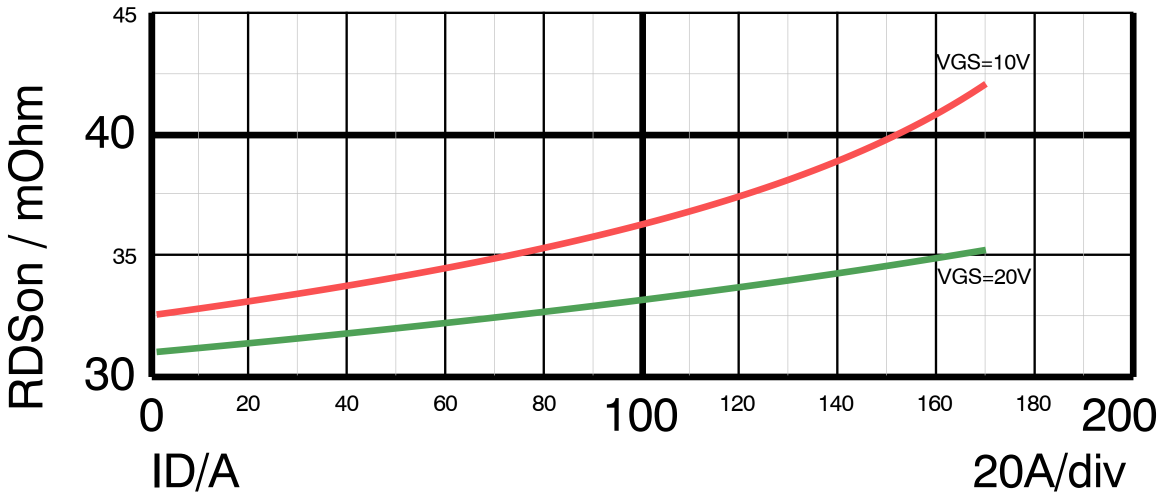
We connected a current source between the drain and source pin.

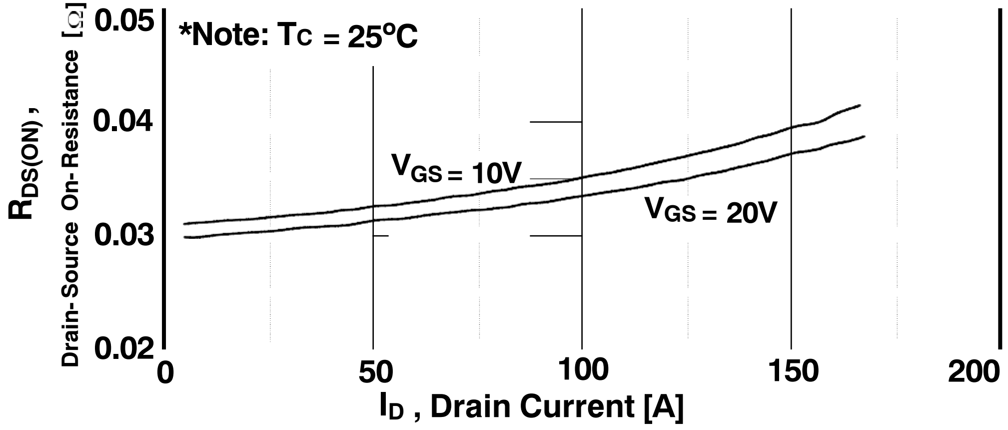
To calculate the $R_{DS(on)}$, we use an arbitrary source to make the division between drain-to-source voltage by the drain current. We obtain a voltage directly proportional to the "on" resistance.

We add a "X-Y Probe" pseudo-component to pre-set the output graph.



$R_{DS(ON)}$ vs Drain Current - Results



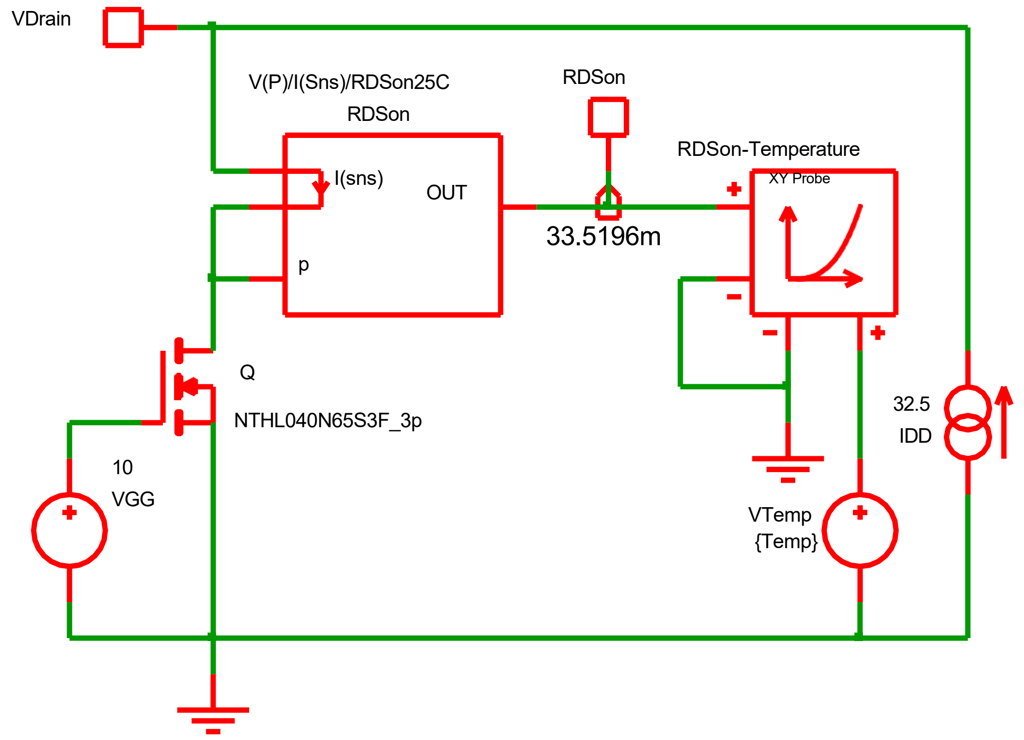


We can see a slight difference between simulation with the NTHL040N65S3F model and the data sheet curve.

In fact, for a 100-A I_D and a 10-V V_{GS}, the difference is almost 1.25 mΩ.
This corresponds to a 3% relative difference.

This is acceptable.

$R_{DS(ON)}$ vs Temperature - Schematic



We use almost the same schematic,

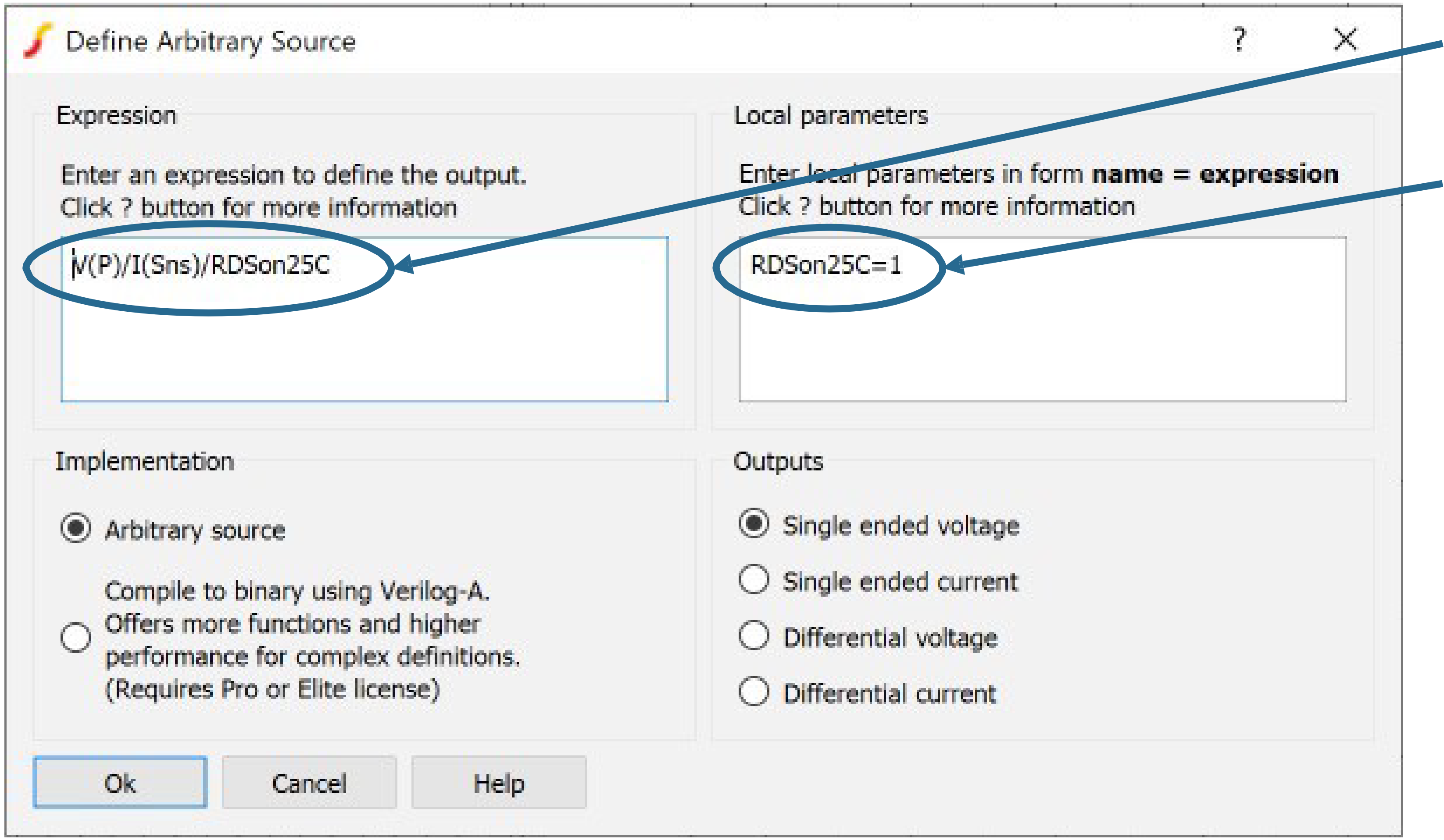
Except :

The current is fixed at 32.5 A,

A voltage source to represent the ambient temperature,

We add a “Bus annotation marker” to have the $R_{DS(on)}$ at 25°C.

$R_{DS(ON)}$ vs Temperature – Setup 1



Expression

Enter an expression to define the output.
Click ? button for more information

V(P)/I(Sns)/RDSon25C

Local parameters

Enter local parameters in form **name = expression**
Click ? button for more information

RDSon25C=1

Implementation

● Arbitrary source

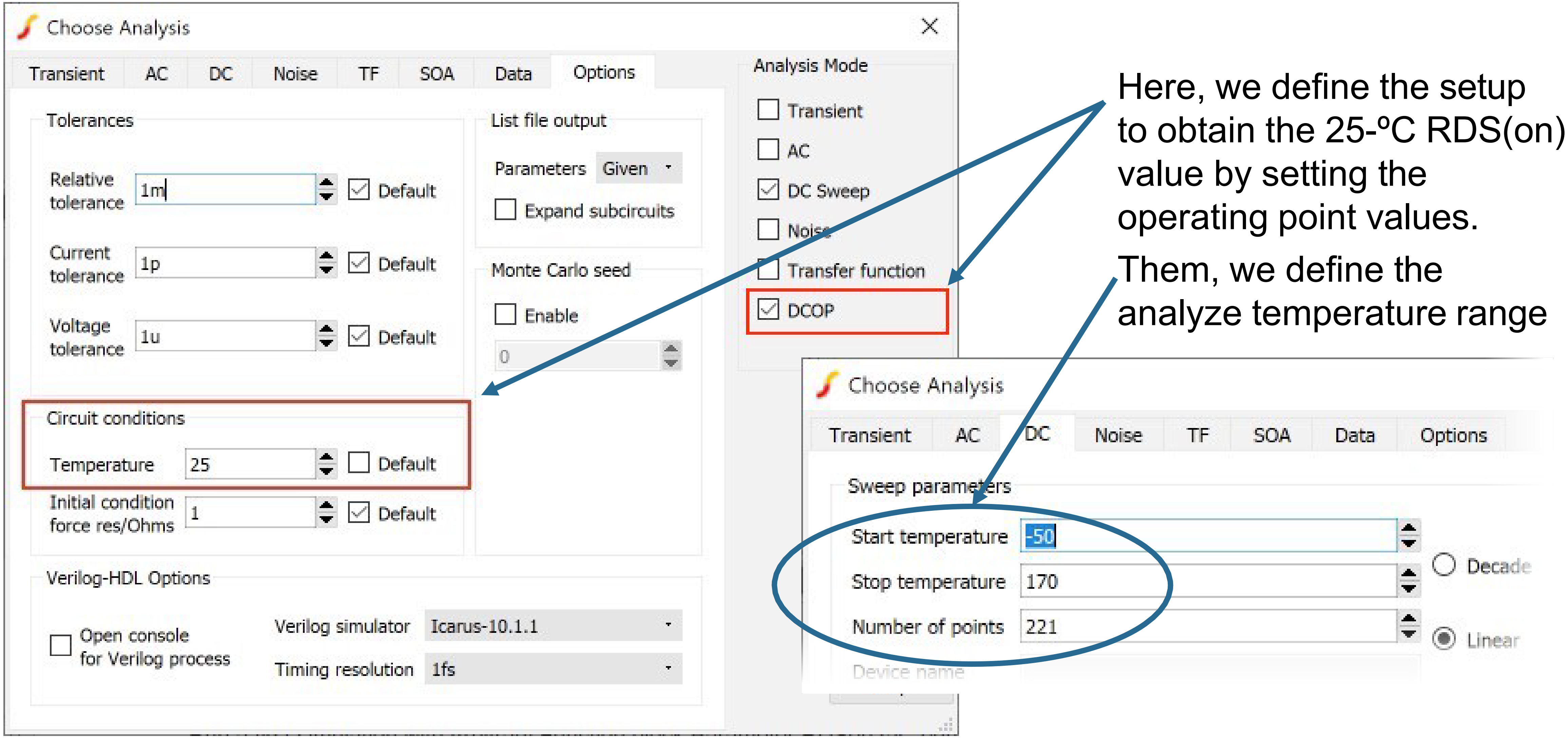
○ Compile to binary using Verilog-A. Offers more functions and higher performance for complex definitions. (Requires Pro or Elite license)

Outputs

● Single ended voltage
○ Single ended current
○ Differential voltage
○ Differential current

Here is the arbitrary function definition with the parameter used to normalize the curve

$R_{DS(ON)}$ vs Temperature – Setup 2



Choose Analysis

Tolerances

Relative tolerance: 1m ☑ Default

Current tolerance: 1p ☑ Default

Voltage tolerance: 1u ☑ Default

Circuit conditions

Temperature: 25 ☐ Default

Initial condition force res/Ohms: 1 ☑ Default

List file output

Parameters: Given

☐ Expand subcircuits

Monte Carlo seed

☐ Enable

0

Verilog-HDL Options

☐ Open console for Verilog process

Verilog simulator: Icarus-10.1.1

Timing resolution: 1fs

Analysis Mode

☐ Transient

☐ AC

☑ DC Sweep

☐ Noise

☐ Transfer function

☑ DCOP

Choose Analysis

Sweep parameters

Start temperature: -50

Stop temperature: 170

Number of points: 221

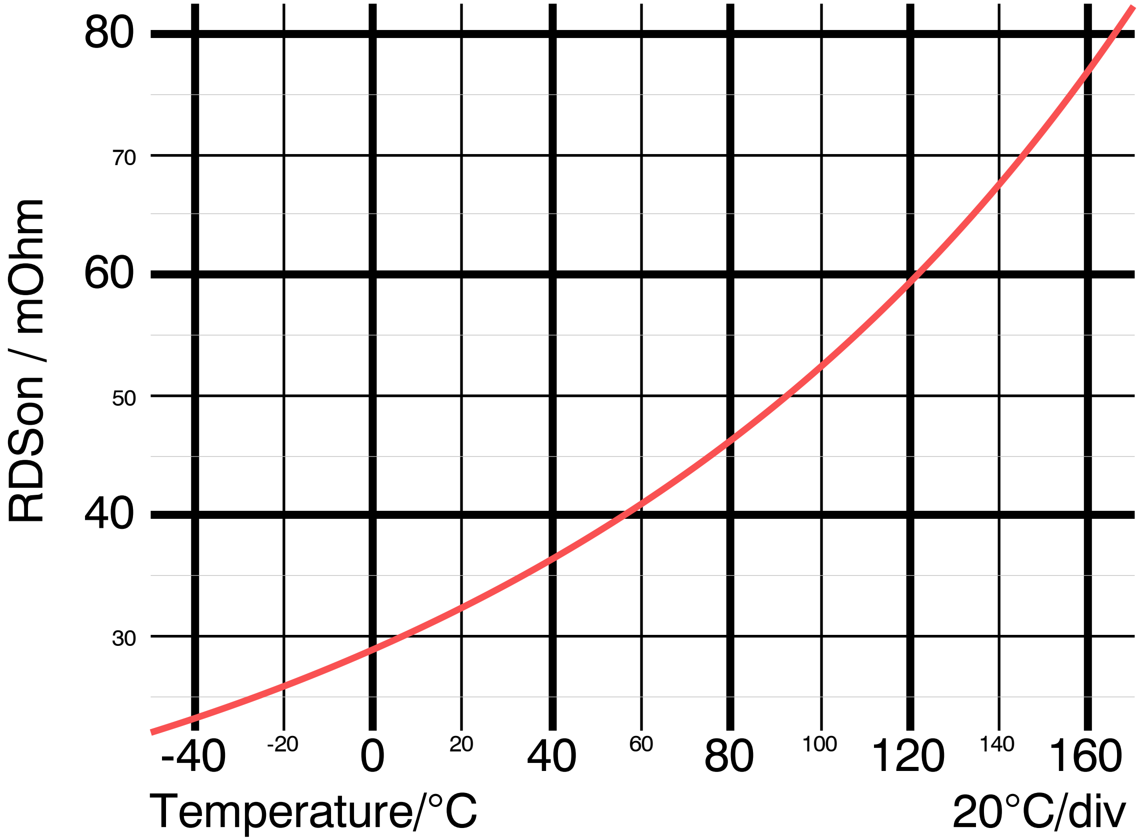
○ Decade

● Linear

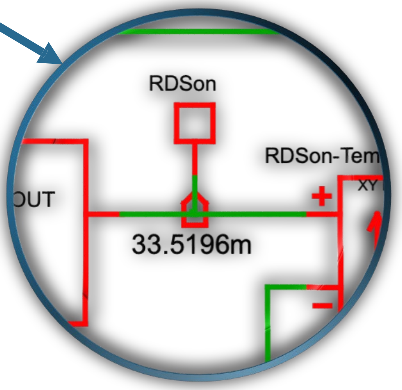
Here, we define the setup to obtain the 25-ºC RDS(on) value by setting the operating point values.

Them, we define the analyze temperature range

$R_{DS(ON)}$ vs Temperature – Results

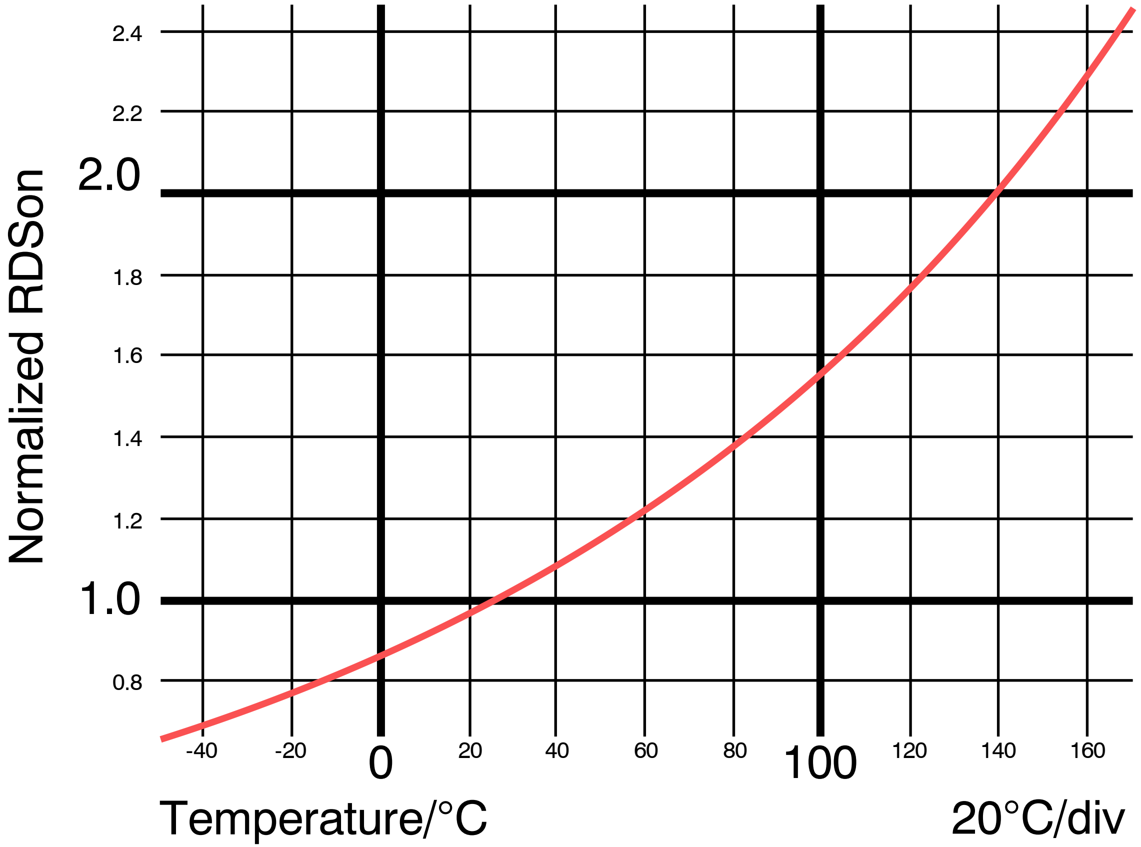


On the schematic, we can read the 25−ºC value for the $R_{DS(on)}$.



This value will be used as parameter value for arbitrary function for the next simulation.

$R_{DS(ON)}$ vs Temperature – Results Normalized

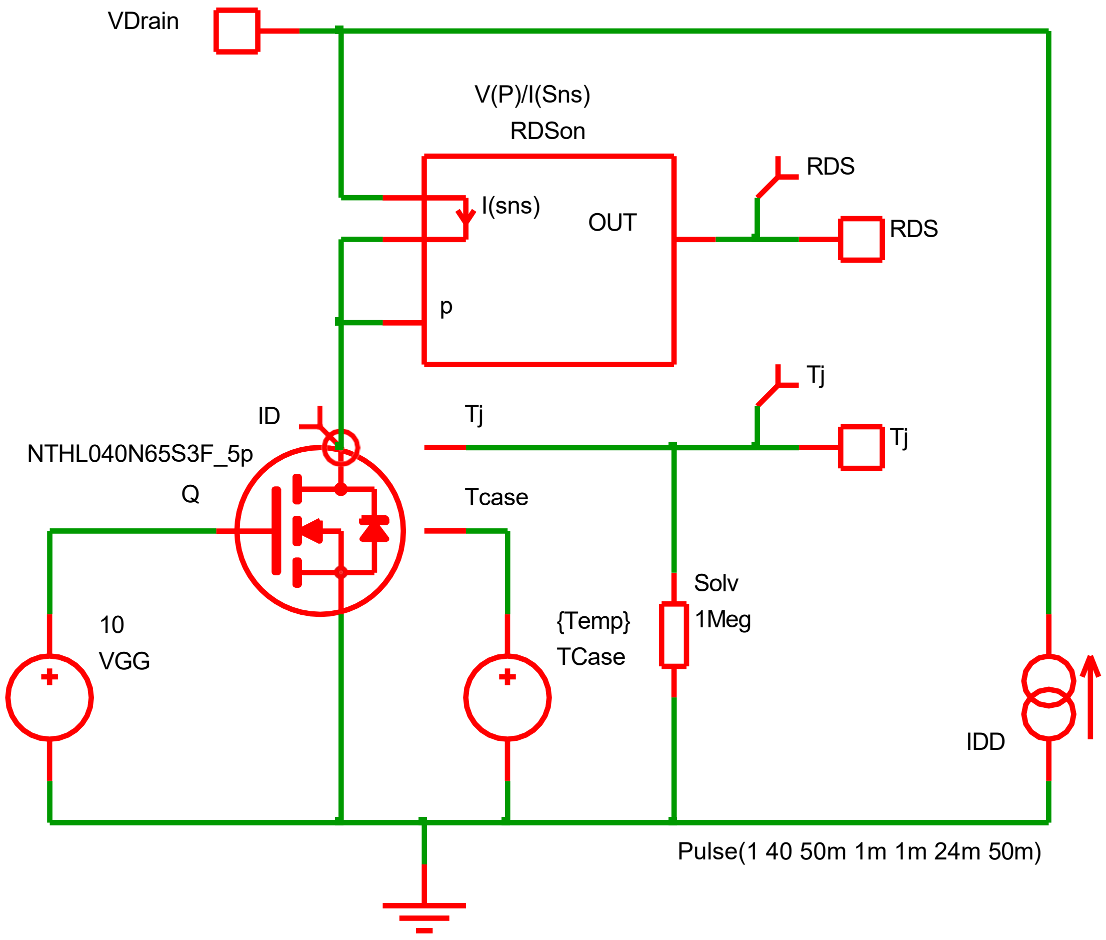


Around 140ºC,
the $R_{DS(on)}$ value is twice
the 25-ºC value.

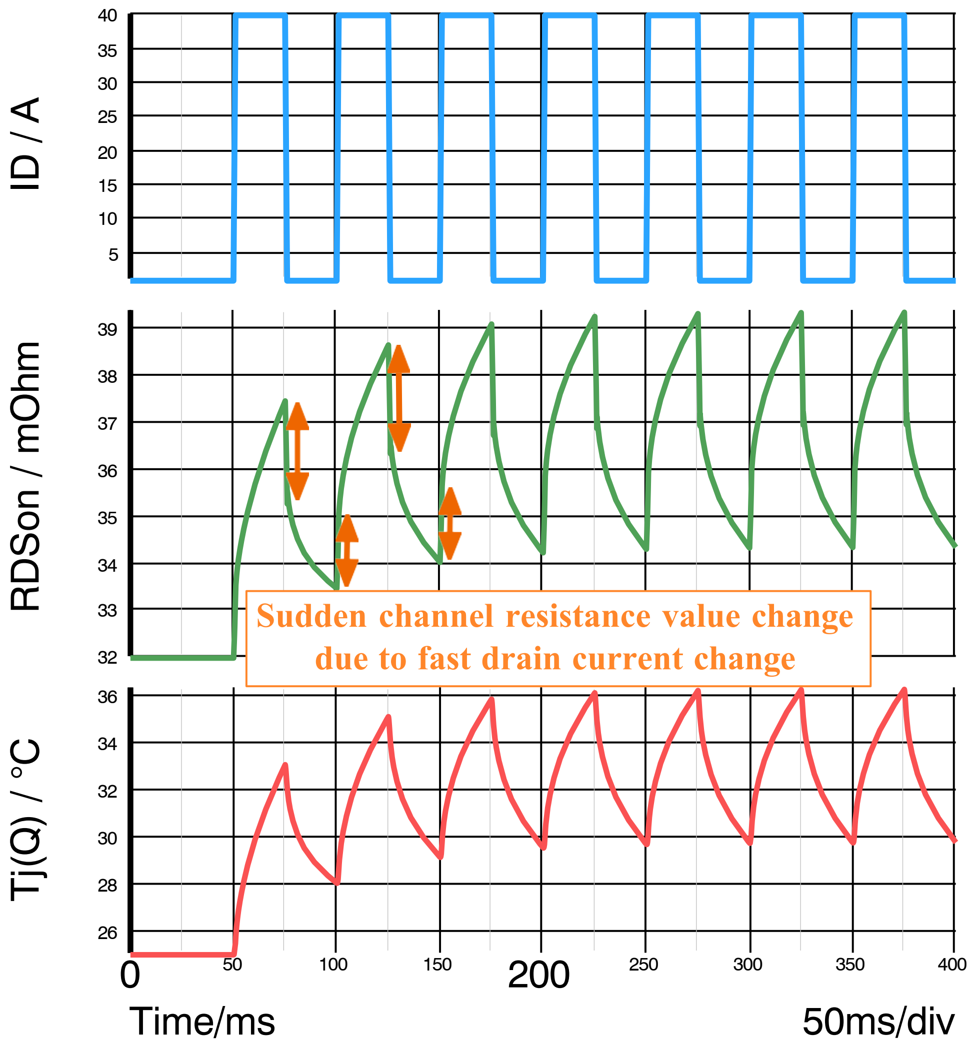
$R_{DS(ON)}$ vs Time - Schematic

The 5-pin models are very useful when you want to know the junction temperature behavior depending on the mission profile.

Here, we analyze a low frequency switching schematic and the impact on junction temperature.



$R_{DS(ON)}$ vs Time - Results



Sudden channel resistance value change due to fast drain current change

We can notice the sudden change in the channel resistance when the drain current changes rapidly from 1 A to 40 A and backward. This phenomenon was predicted by the curve Channel resistance vs Drain current.

We also see the effect of the self-heating or cooling of the device itself during the plateau phase of the current (1 A and/or 40 A).

The system is stable after a 200-ms transient. The maximum junction temperature is 36 °C and the minimum 30 °C. The junction temperature oscillates between those two values.

Transfer characteristic

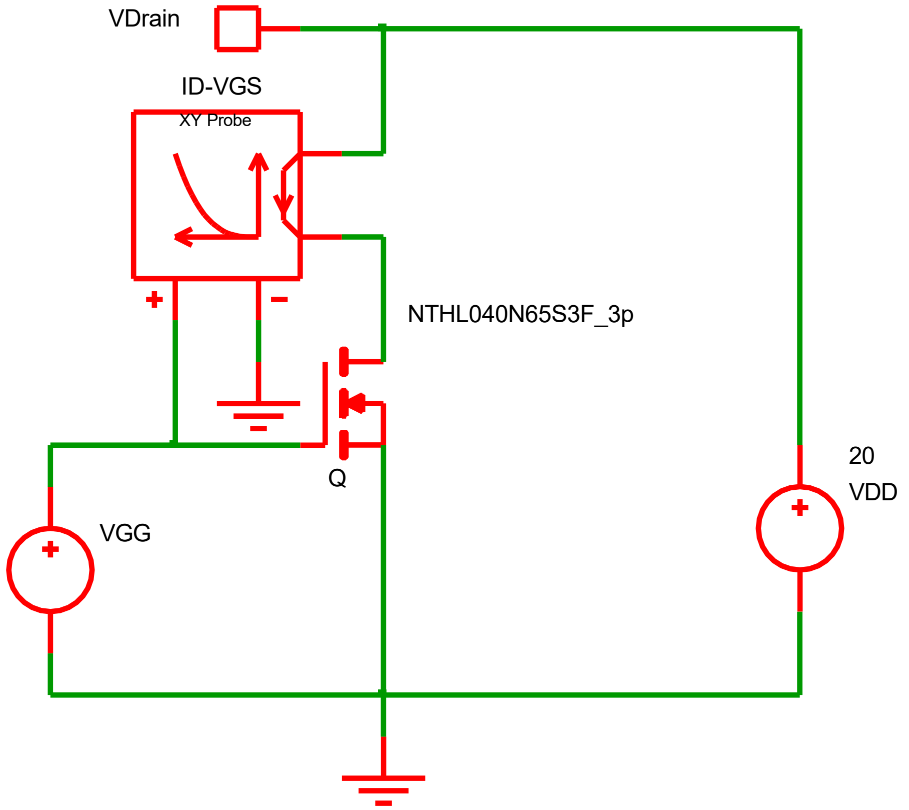
Drain Current vs Gate-to-Source Voltage

Transfer Characteristic - Schematic

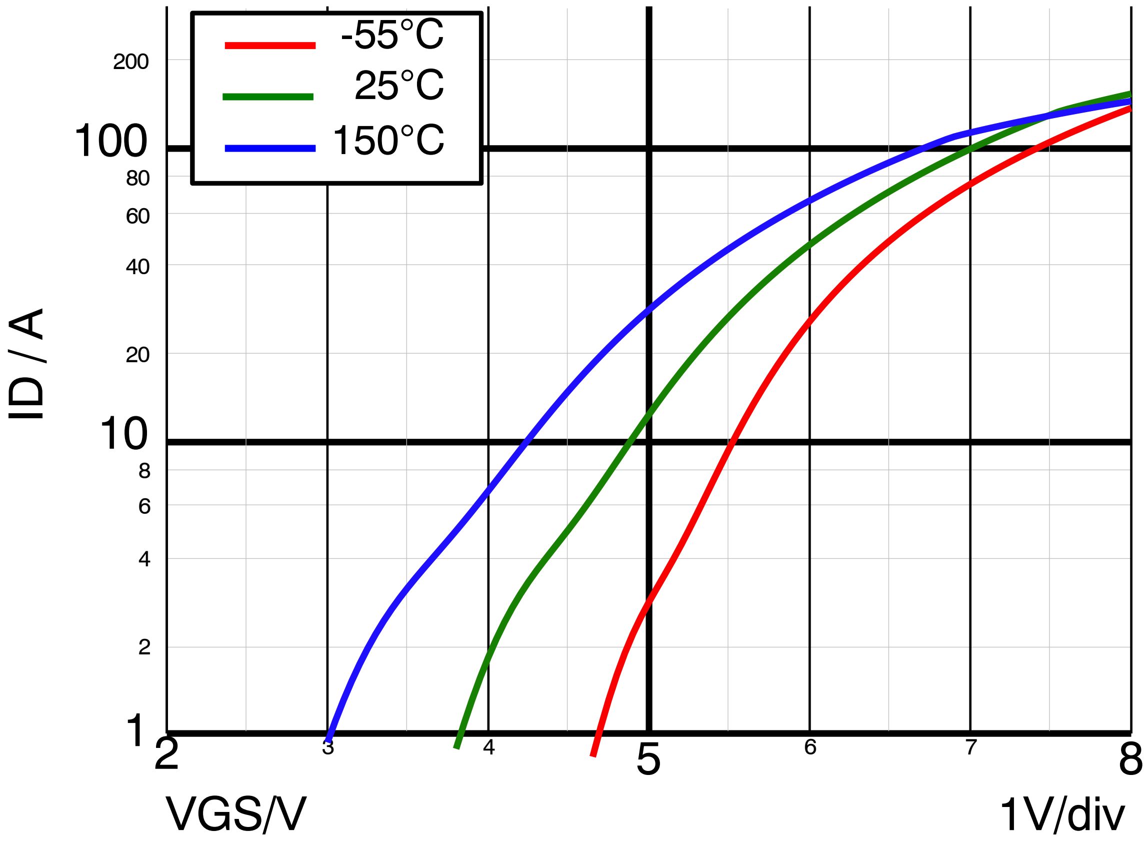
The transfer characteristic shows how the drain current changes with the gate-to-source voltage.

The simulation was done with a 20-V drain-to-source voltage and for various temperature values.

We will use a X-Y probe to plot the transfer characteristic and the temperature will be set via the “Multi-Step” simulation list.



Transfer Characteristic - Results



We can see the difference for the gate threshold in function of temperature.

Output Capacitor

Small Signal, Effective, Energy related or Charge related

Output Capacitor

For switching application, the output capacitor called C_{oss} defined as

$$C_{!""} = (\mathrm{C}_{\#\$} + C_{\%\#}) \quad @V_{\%\$} = 0\ \mathrm{V}$$

is an important parameter as it has an impact on the transistor switching losses. In fact, every time the MOSFET turn on, the energy stored in output capacitance is discharged and lost in the transistor. The lower C_{oss} is the better. C_{oss} is a non-linear capacitance and highly depends on the drain-to-source voltage.

There are 3 to 4 types of output-capacitance values found in the specification.

The capacitance types are called:

Small-signal value,

Effective value,

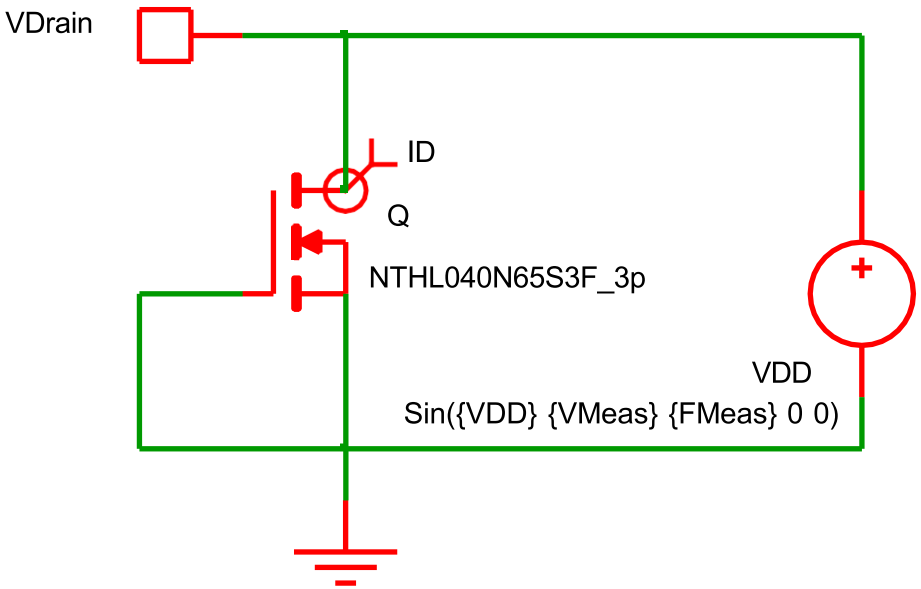
Energy-related value,

Charge-related value.

Output Capacitor Small-signal - Schematic

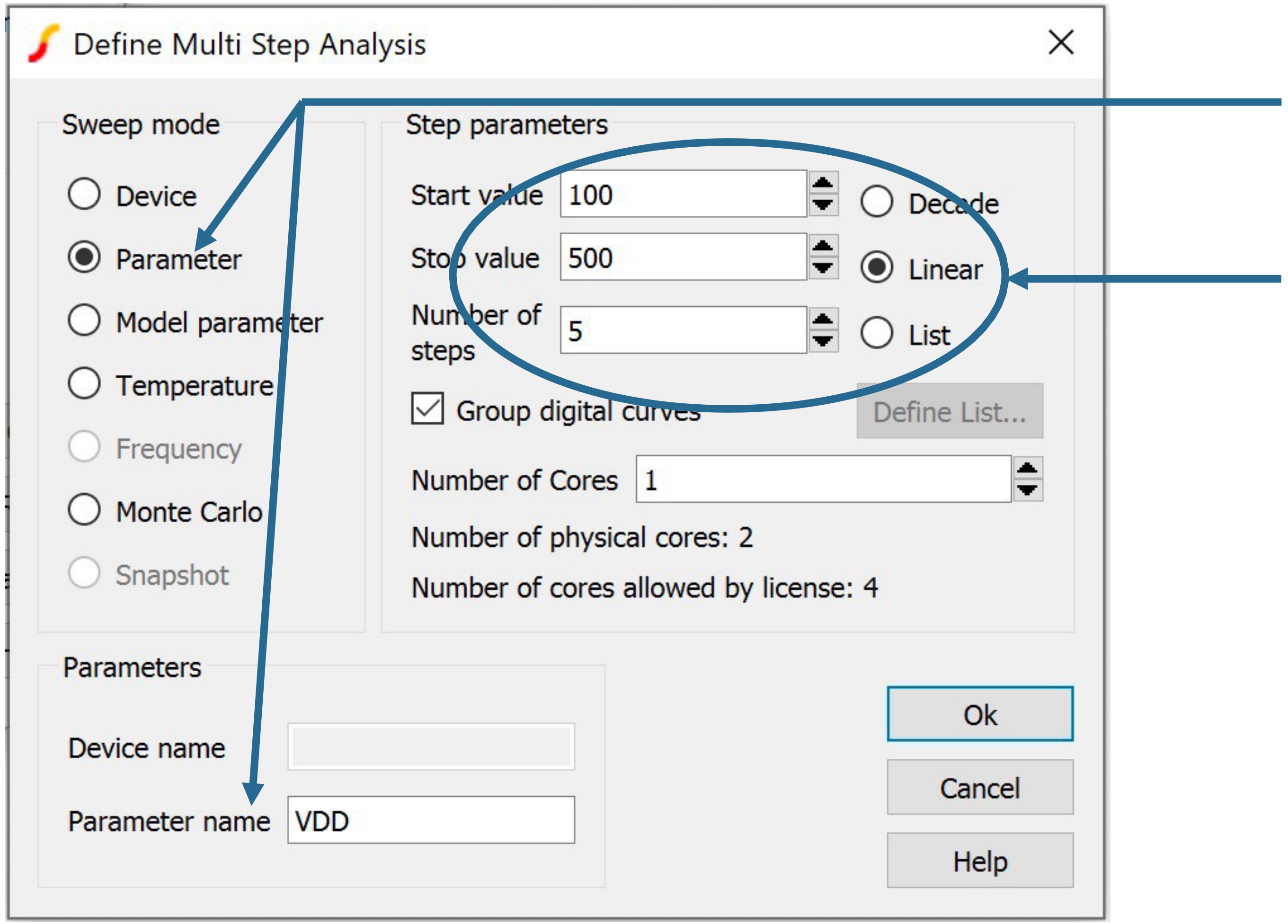
For the signal signal value, we will use the following equation :

$$C_{!""} = \frac{\&}{'(\times *} \times \frac{(\#+,-.\ \ /0++1.2)_{!"\#\$-\&'-!"\#\$}}{(\#+,-.-2!-\$!0+41\ 5!62,71)_{!"\#\$-\&'-!"\#\$}}$$



We will use 20-mV peak-to-peak sinusoidal voltage source with frequency equal to 1 MHz in series with the drain-to-source continuous voltage.

Output Capacitor Small-signal – Setup



Define Multi Step Analysis

Sweep mode
- ○ Device
- ● Parameter
- ○ Model parameter
- ○ Temperature
- ○ Frequency
- ○ Monte Carlo
- ○ Snapshot

Step parameters

Start value: 100

Stop value: 500

Number of steps: 5

○ Decade ● Linear ○ List

☑ Group digital curves

Number of Cores: 1

Number of physical cores: 2

Number of cores allowed by license: 4

Parameters

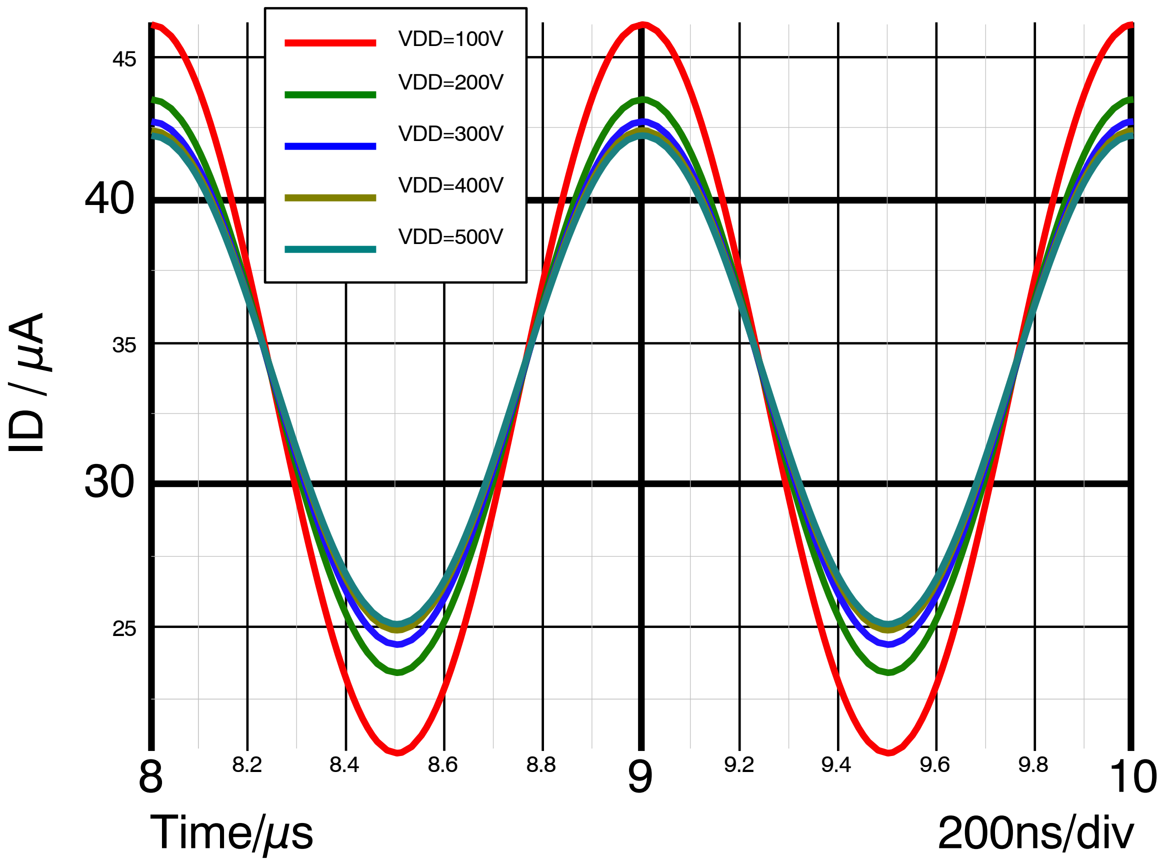
Device name:

Parameter name: VDD

{VDD} is a parameter for the drain-to-source voltage continuous value.

We will sweep this value

Output Capacitor Small-signal – First Results



We show here only the two last milli-seconds.

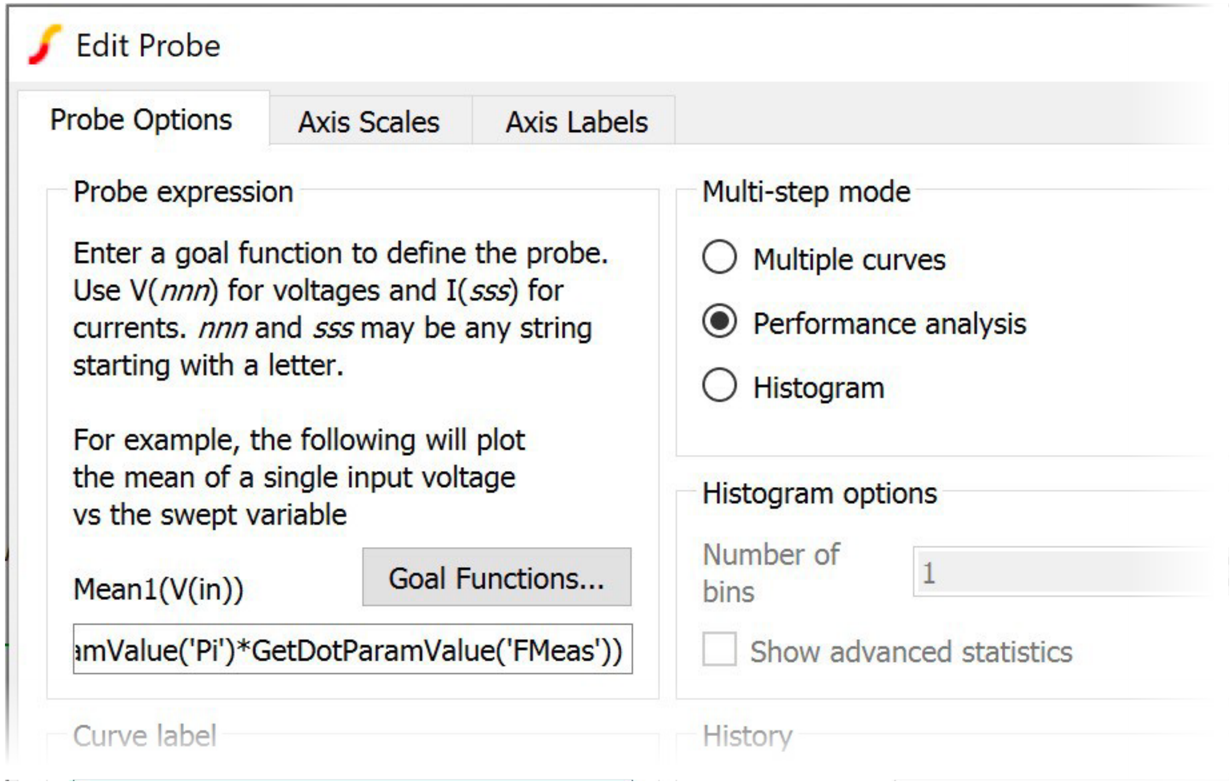
We can notice the continuous current offset (around 32 µA) corresponding to the Drain-to-Source leakage current.

The peak-to-peak value depends on the continuous Drain-to-Source voltage.

So, the output capacitor also…

Output Capacitor Small-signal – Setup 2

To calculate the output capacitor, we use an arbitrary-function probe with the previous C_{OSS} equation.



Probe expression

Enter a goal function to define the probe. Use V(*nnn*) for voltages and I(*sss*) for currents. *nnn* and *sss* may be any string starting with a letter.

For example, the following will plot the mean of a single input voltage vs the swept variable

Mean1(V(in))

amValue('Pi')*GetDotParamValue('FMeas'))

Multi-step mode

○ Multiple curves
● Performance analysis
○ Histogram

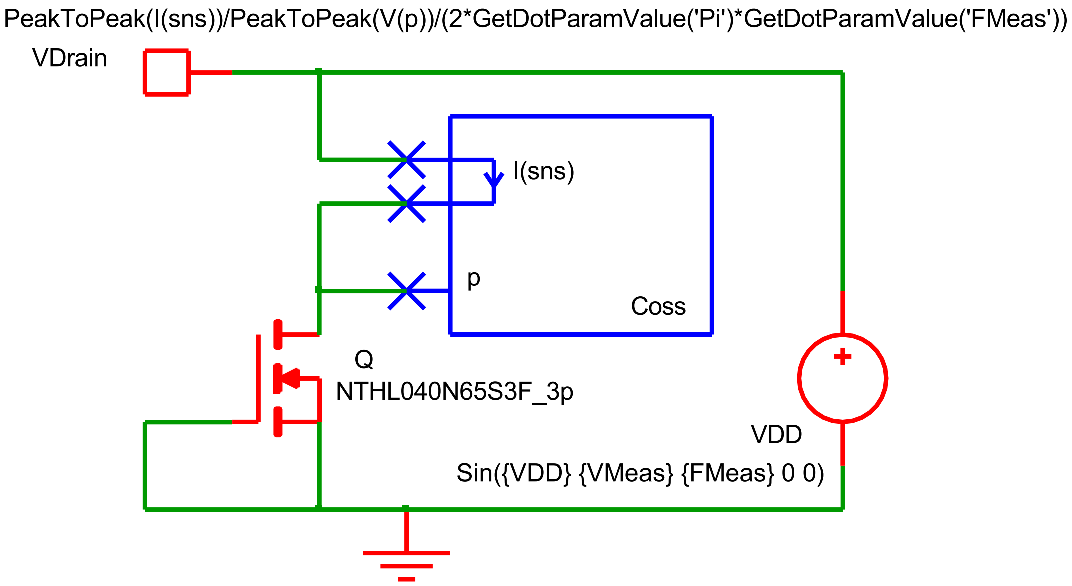
Histogram options

Number of bins: 1

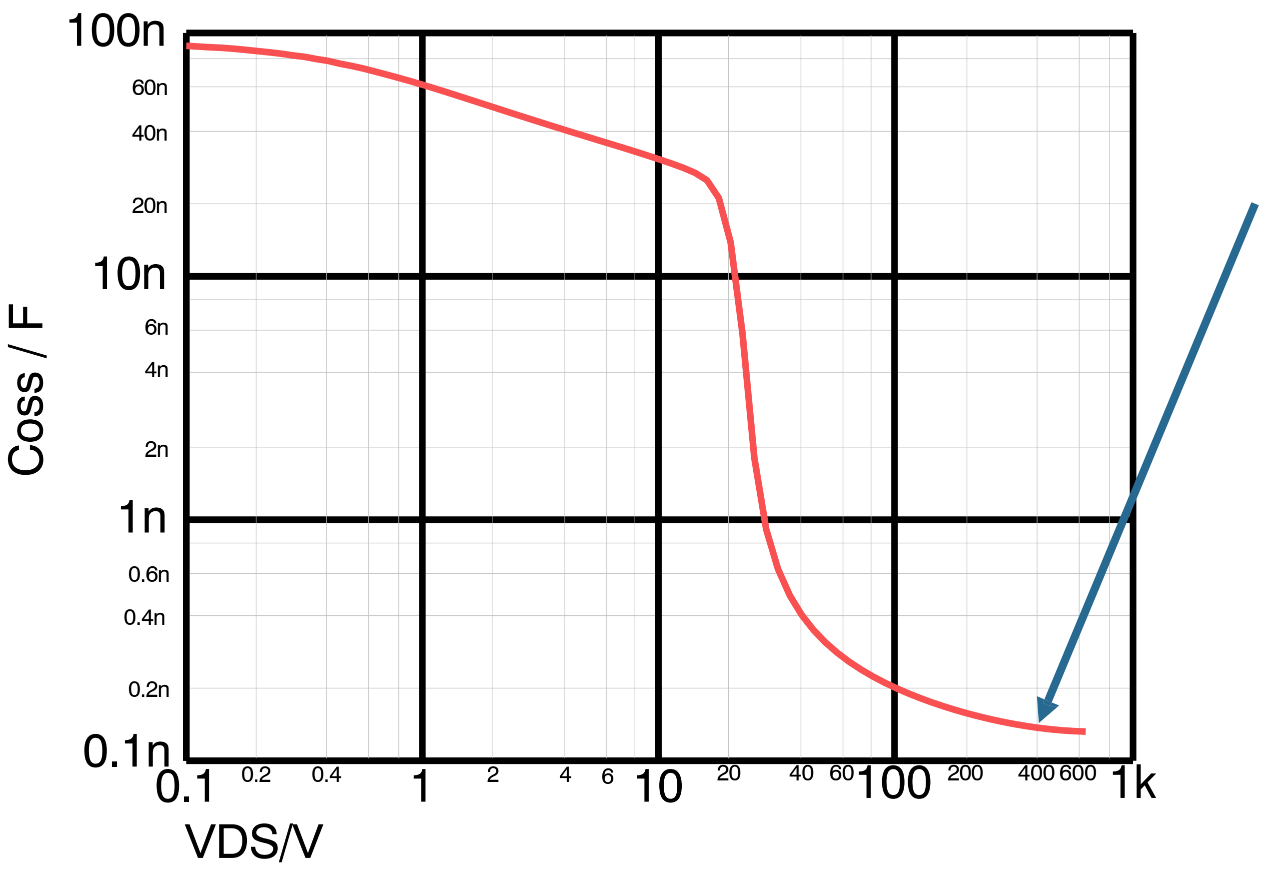
☐ Show advanced statistics

Curve label

History



Output Capacitor Small-signal – Final Results



These measurement are meet the datasheet results. As an example, for a drain-to-source equal to 400 V, we measured a C_{OSS} value of 142 pF in the figure.

This value matches with 140 pF given in the datasheet.

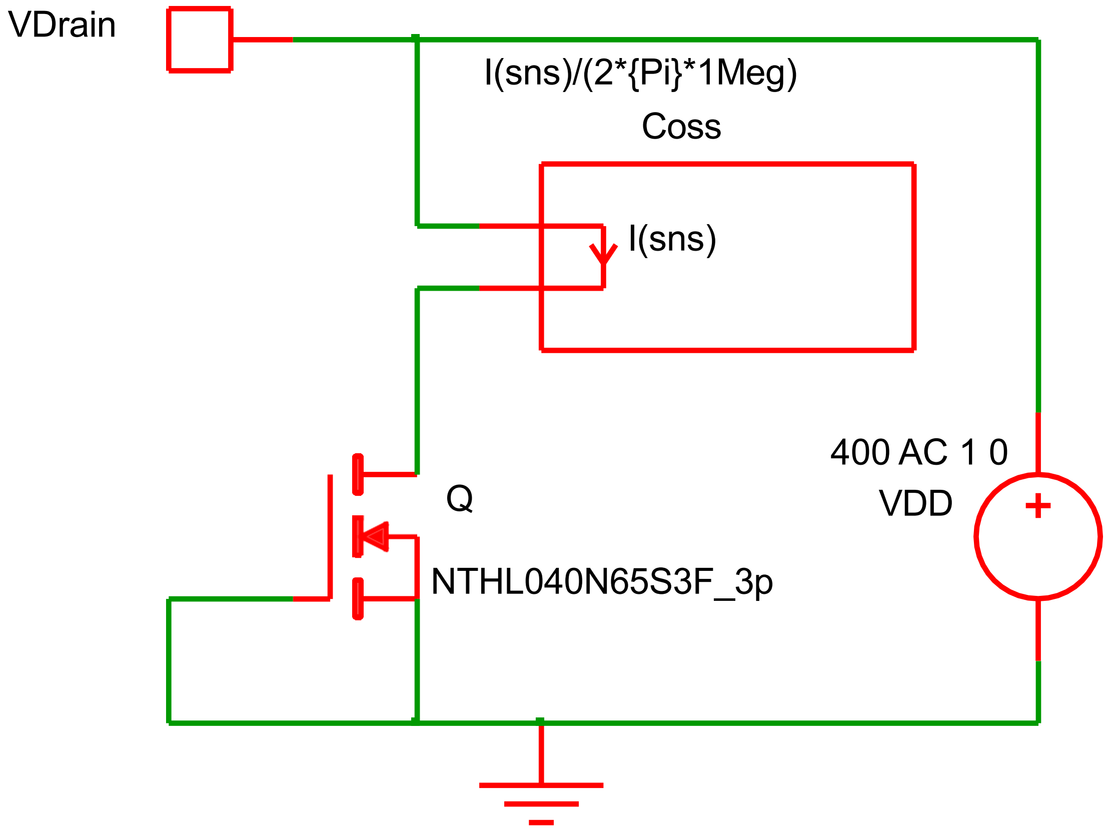
Output Capacitor Small-signal – Schematic / 2nd Method

In small-signal simulation, SIMetrix offers the possibility to sweep other parameters than frequency. Here, we will set the frequency to 1 MHz and sweep the drain-to-source dc voltage.

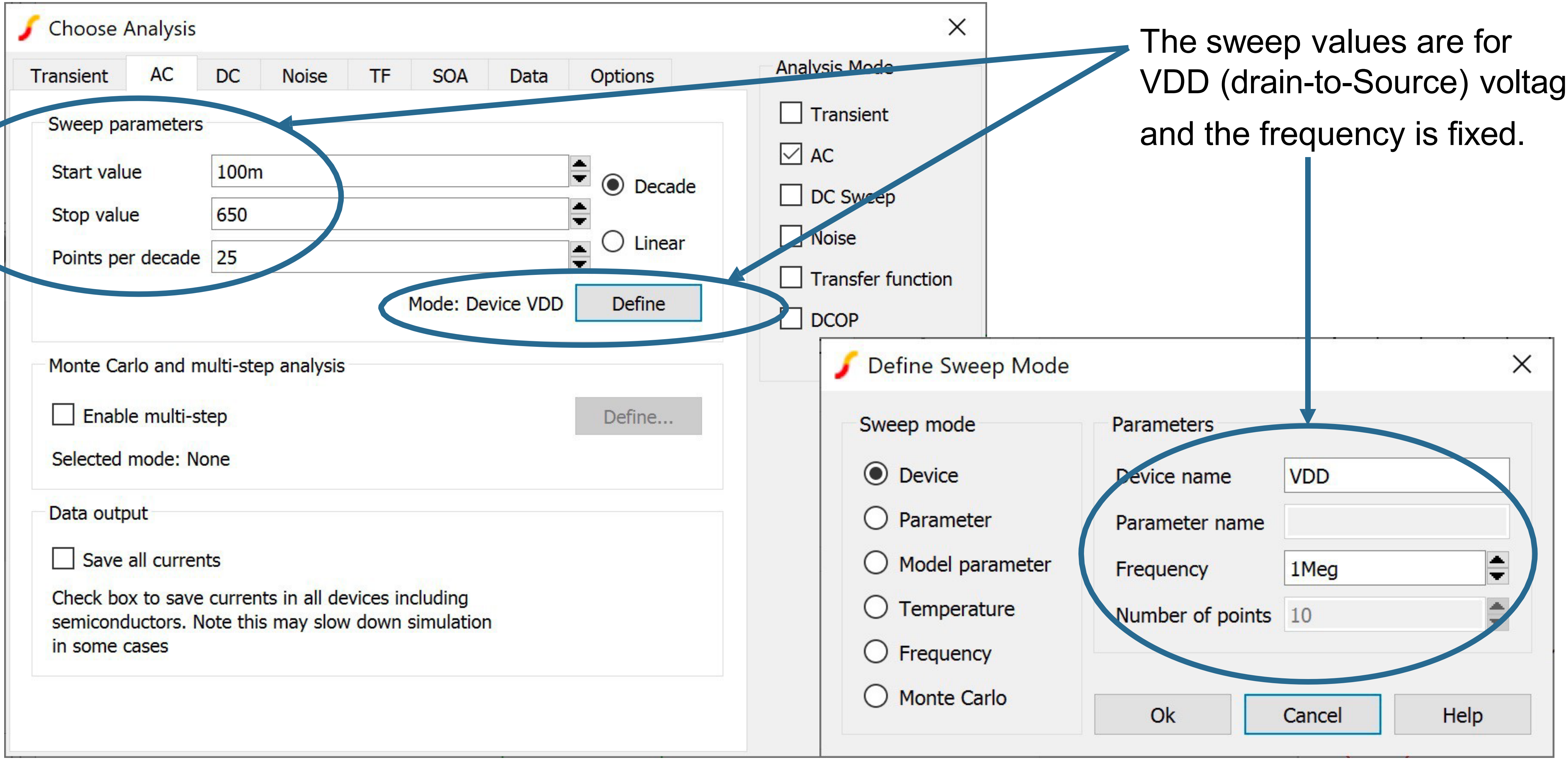
We will use the following equation to get C_{OSS} values :

$$C_{!""} = \frac{\#_{(\#)}}{\$\%\times'\times(_{(*\#)}}$$

As the small signal is 1 V (0 dBV), it doesn't count in the equation.



Output Capacitor Small-signal – Setup / 2nd Method



Choose Analysis

Sweep parameters

Start value: 100m

Stop value: 650

Points per decade: 25

● Decade

○ Linear

Mode: Device VDD

Monte Carlo and multi-step analysis

☐ Enable multi-step

Selected mode: None

Data output

☐ Save all currents

Check box to save currents in all devices including semiconductors. Note this may slow down simulation in some cases

Analysis Mode

☐ Transient

☑ AC

☐ DC Sweep

☐ Noise

☐ Transfer function

☐ DCOP

Define Sweep Mode

Sweep mode

● Device

○ Parameter

○ Model parameter

○ Temperature

○ Frequency

○ Monte Carlo

Parameters

Device name: VDD

Parameter name:

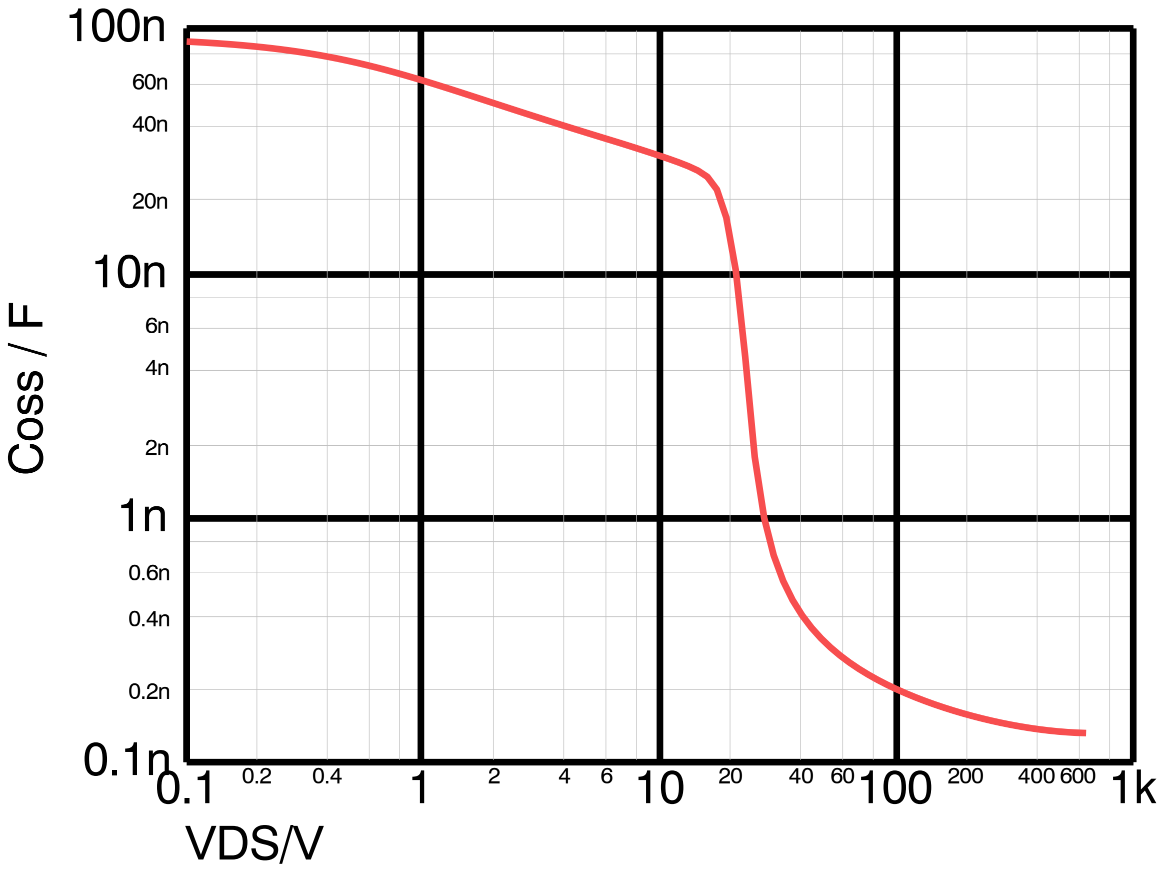
Frequency: 1Meg

Number of points: 10

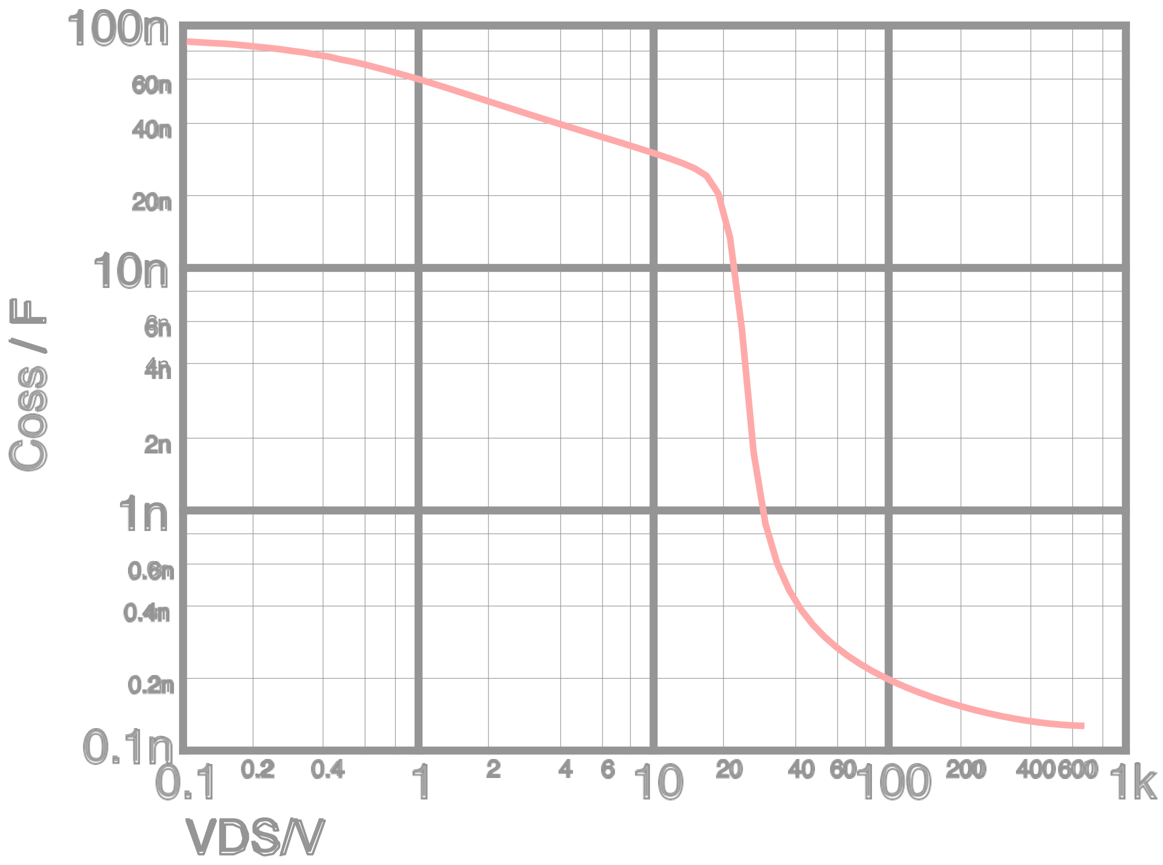
The sweep values are for VDD (drain-to-Source) voltage and the frequency is fixed.

Output Capacitor Small-signal – Results / 2nd Method

- **2nd Method**



- **1st Method**



Output Effective Capacitor - Definition

The effective-capacitor value is defined as the equivalent linear capacitor storing the same amount of charge/energy with a voltage source equal to breakdown voltage value and a 100-k O series resistor to charge the output capacitor.

This value can be used to calculate the switching time in resonant topologies.

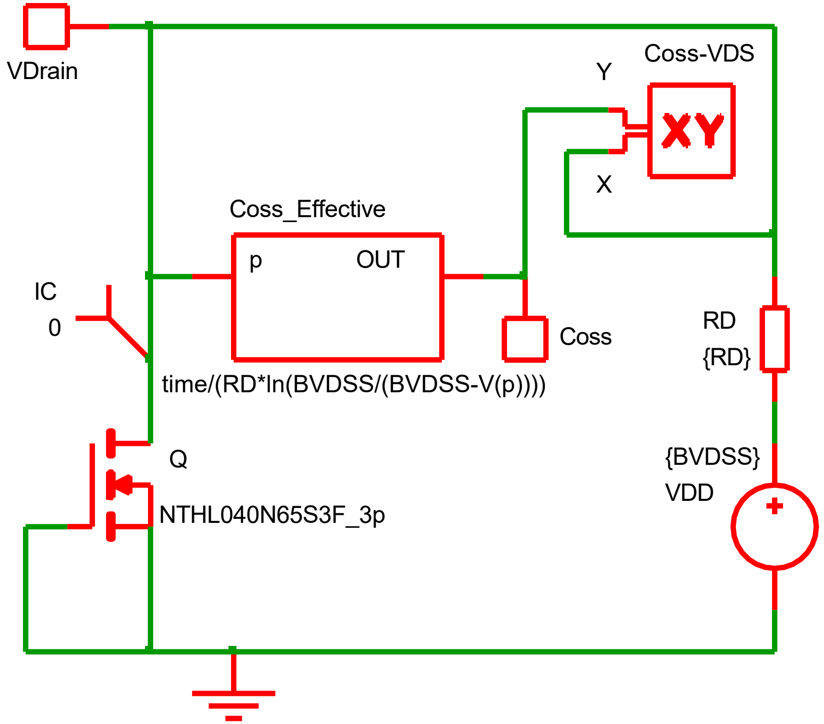
The charging equation for the linear capacitor in this configuration is given by:

$$V_{)*}(Time) = V_{))}\left(1 - e^{\frac{+,-.\,"}{+_{/(\times 1_{2**}}}}\right)$$

Solving this equation to get C_{OSS} gives:

$$C_{,**} = \frac{-./0}{1)\times 23\left(\frac{3_{((}}{3_{((}+3_{(*}(,-.\,"\,)}\right)}$$

Output Effective Capacitor - Schematic



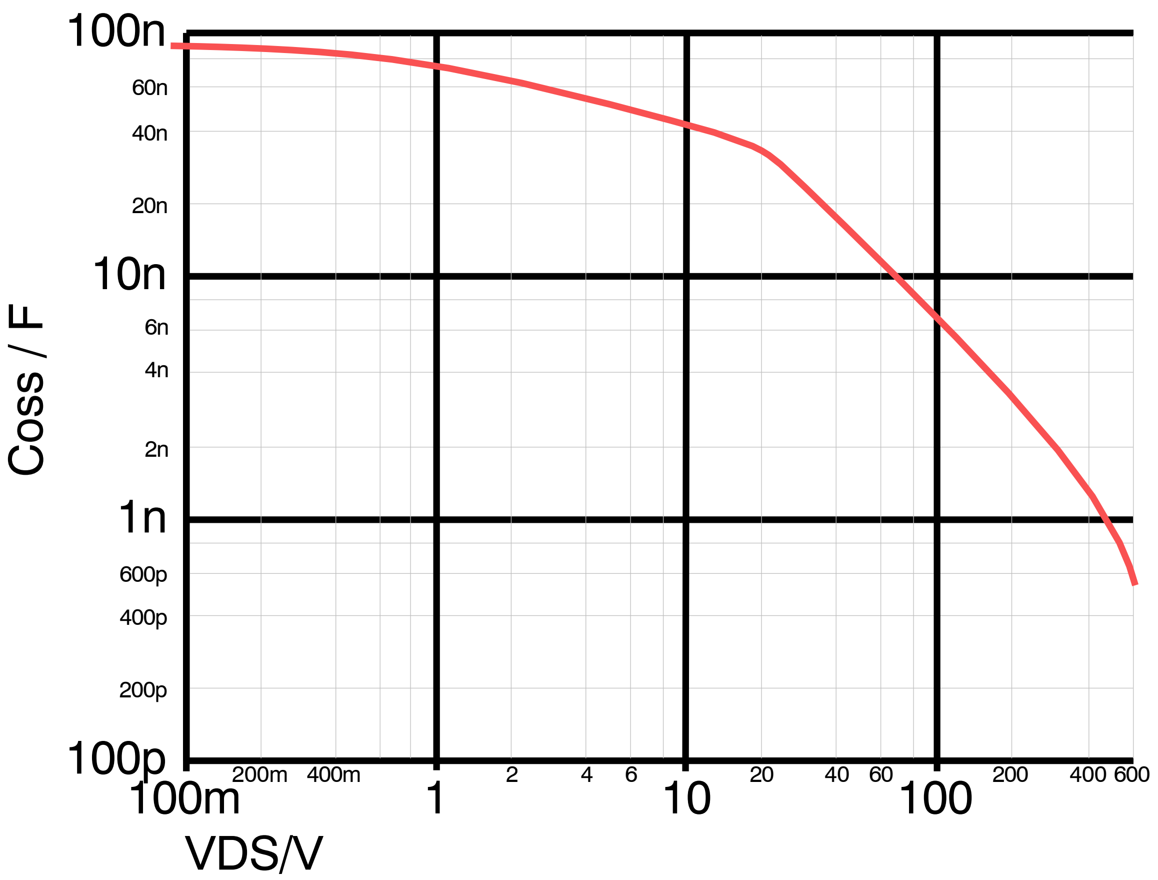
We use an arbitrary function to get the C_{OSS} values and X-Y probe.

The values for the resistor (RD) and the voltage source (VDD) are set with parameters in the “Command Windows” (F11). The values are 100 kν and 650 V respectively.

```
.Param RD=100k
.Param BVDSS=650
```

We set an initial condition for the drain-to-source voltage using the “IC” pseudo-component.

Output Effective Capacitor - Results



In the “Dynamic Characteristics” table, the effective output capacitor is given for a drain-to-source voltage equal to 400 V.

On the side figure, we measure a C_{OSS} value equal to 1305 pF on the simulated curve.

This matches with 1366 pF given in the specification.

Energy-related Output Capacitor - Definition

The energy store in a capacitor is expressed by the following equation:

$$\mathrm{d}W = v(t)\times i(t)\times dt$$

And the final energy for a constant capacitor can be express by the following equation:

$$W = \frac{1}{2}CV^{'}$$

We can extract the capacitor value:

$$C_{8\$\$} = \frac{2\int_{9}^{:-;\ 1} V_{\#\$}(t)\times I_{\#}(t)\times dt}{V_{\#\$}^{'}\ (Time)}$$

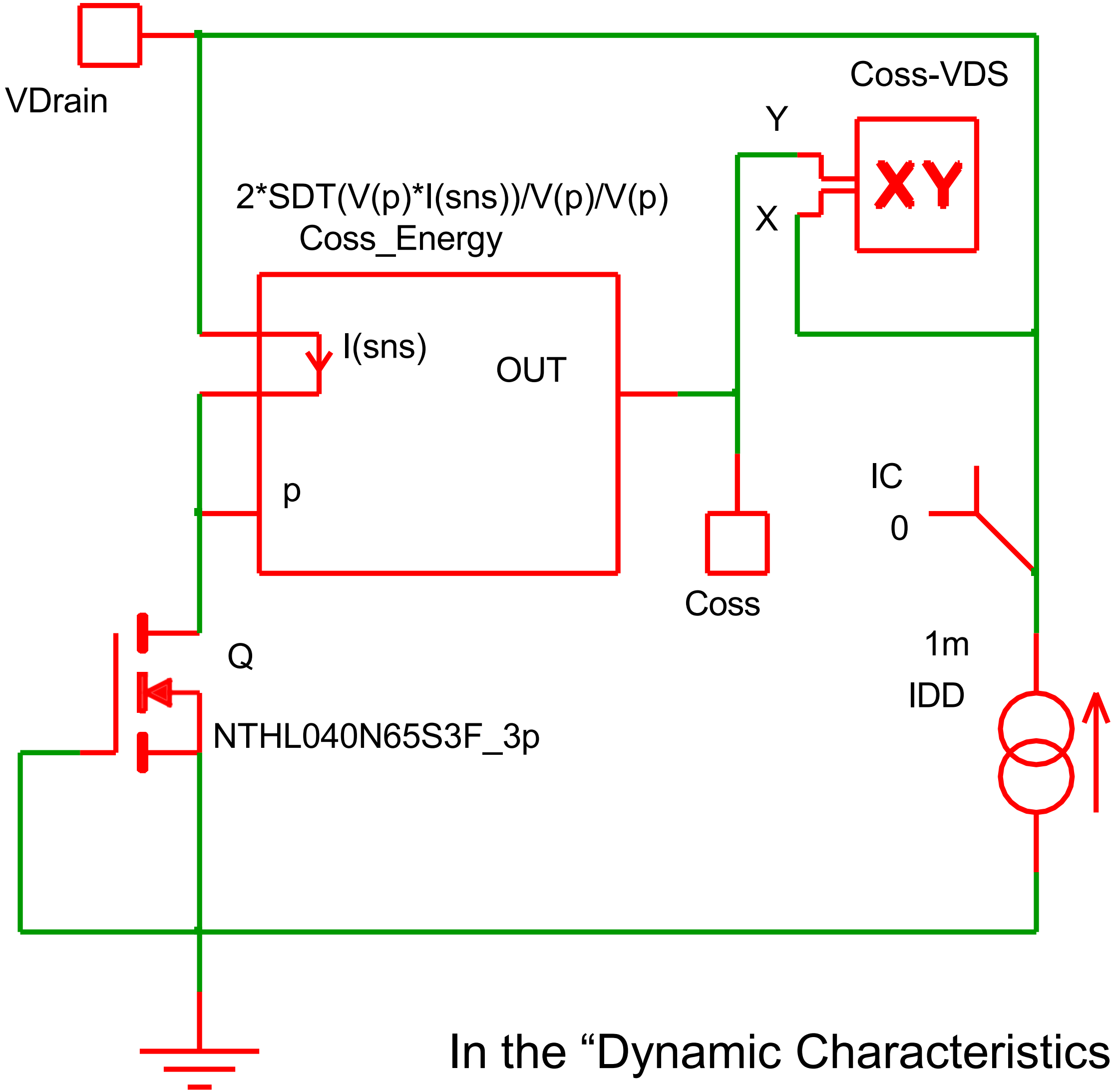
We use a current source to charge the output.

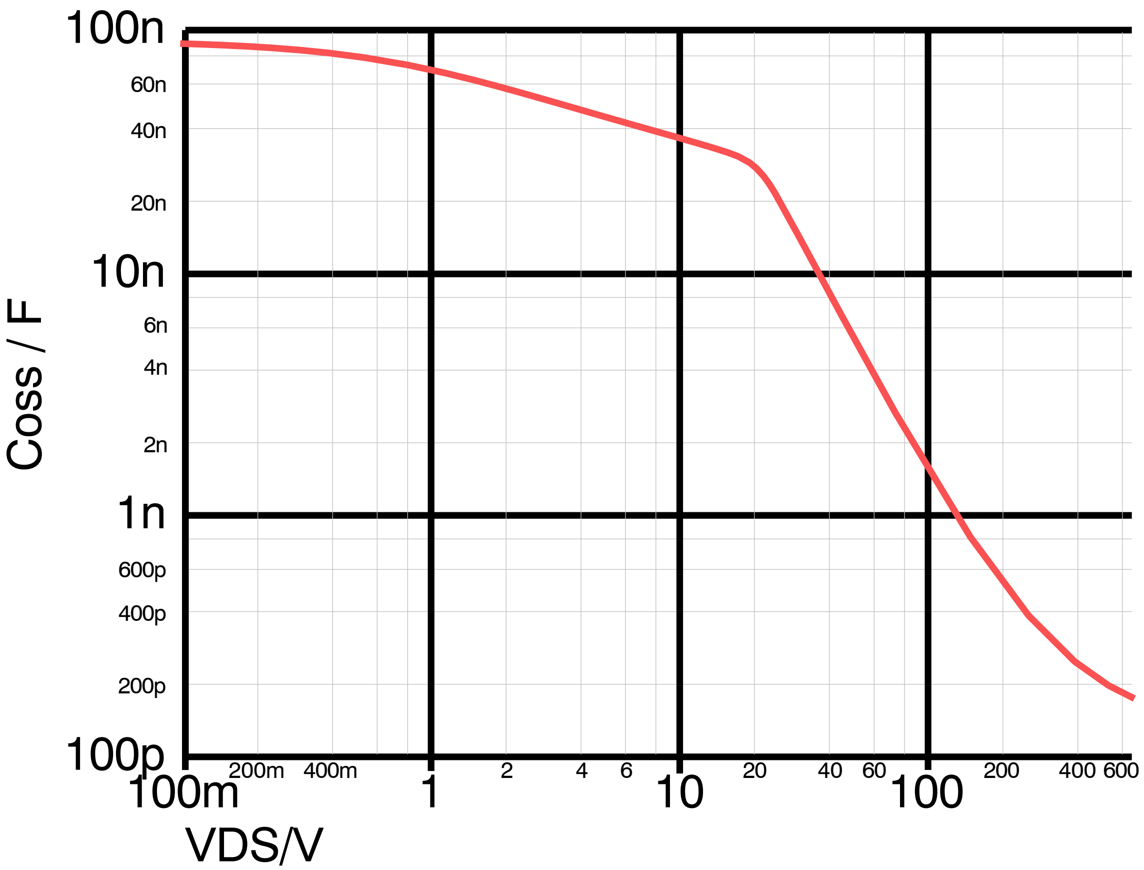
Here also, we use this formula in an arbitrary function to get the C_{OSS} value directly.

We use the integral function “SDT()” to calculate the numerator.

We set an initial condition for the drain-to-source voltage using the “IC” pseudo-component.

Energy-related Output Capacitor – Schematic and Results





In the "Dynamic Characteristics" table, the effective output capacitor is given for a drain-to-source voltage equal to 400 V. We measured a C_{OSS} value equal to 245 pF on the simulated curve above and this matches with 247 pF given in the specification.

Charge-related Output Capacitor - Definition

The charge store in a capacitor is expressed by the following equation:

$$dQ = i(t)\times dt$$

And, the final charge for a constant capacitor can be express by the following equation:

$$Q = CV$$

We can extract the capacitor value:

$$C_{8\$\$} = \frac{\int_{4}^{,\,-.\ \ ''} =_{(}(2)\times>2}{5_{(\,*}(:-;1\ )}$$

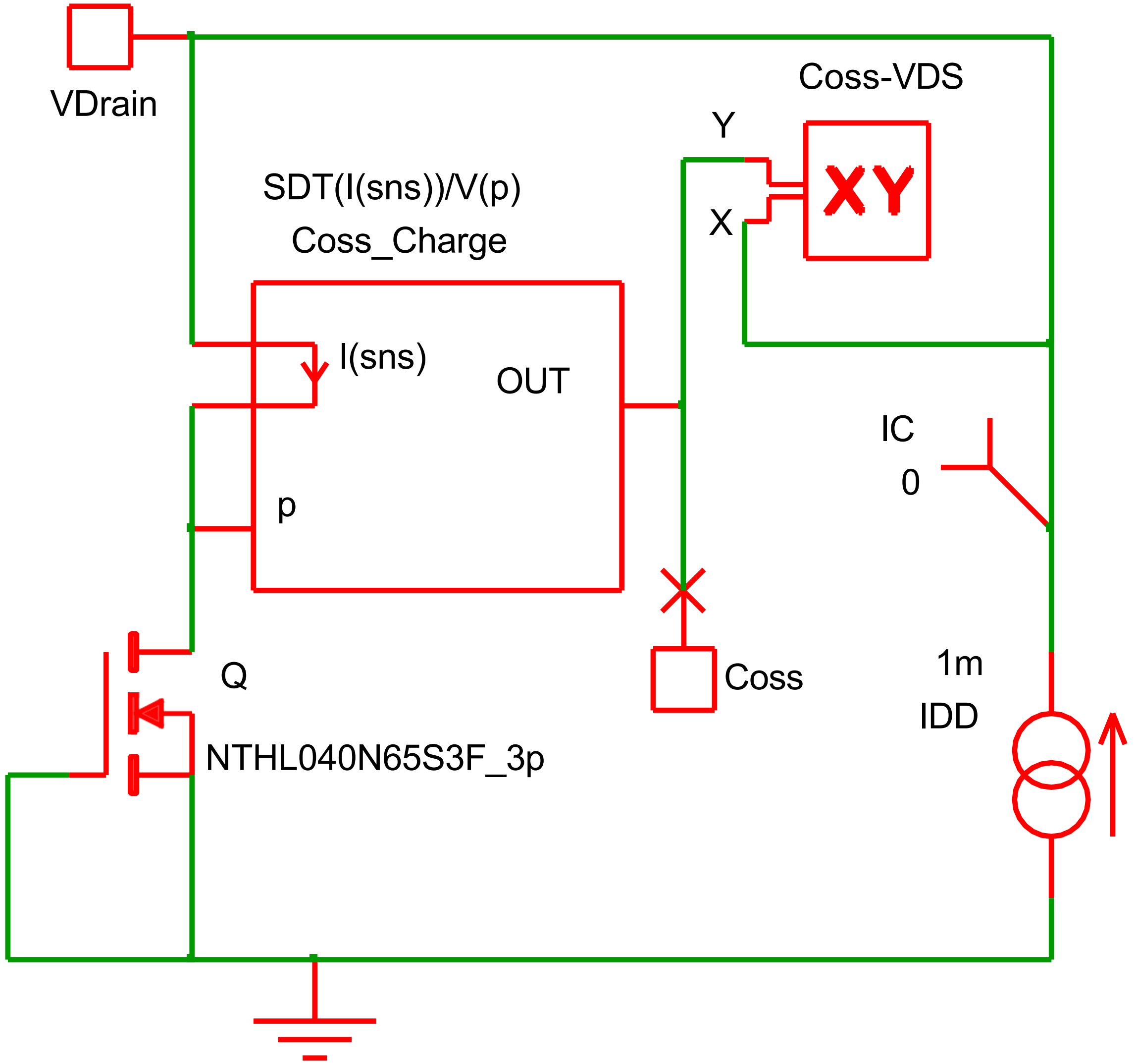
We use almost the same schematic.

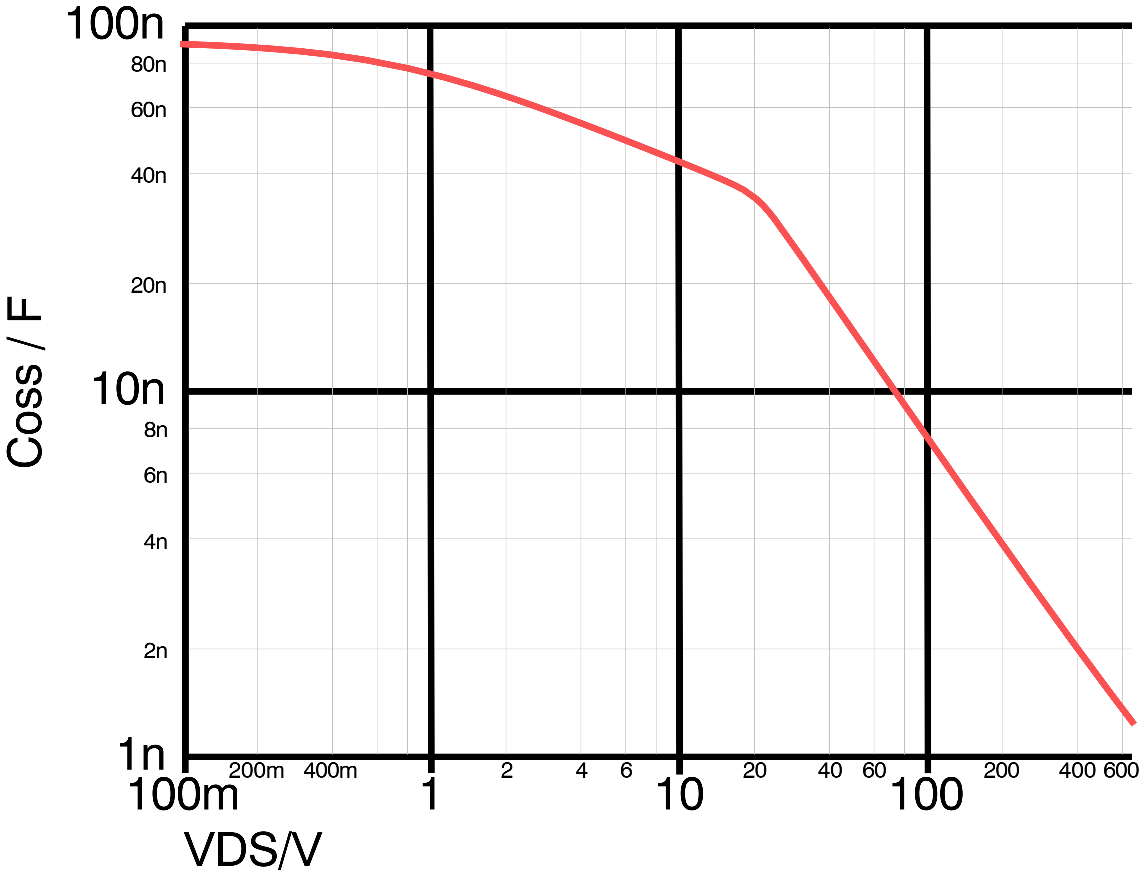
Here also, we use this formula in an arbitrary function to get the C_{OSS} value directly.

We use the integral function “SDT()” to calculate the numerator.

We set an initial condition for the drain-to-source voltage using the “IC” pseudo-component.

Charge-related Output Capacitor – Schematic and Results





Breakdown Voltage (and Drain Leakage Current)

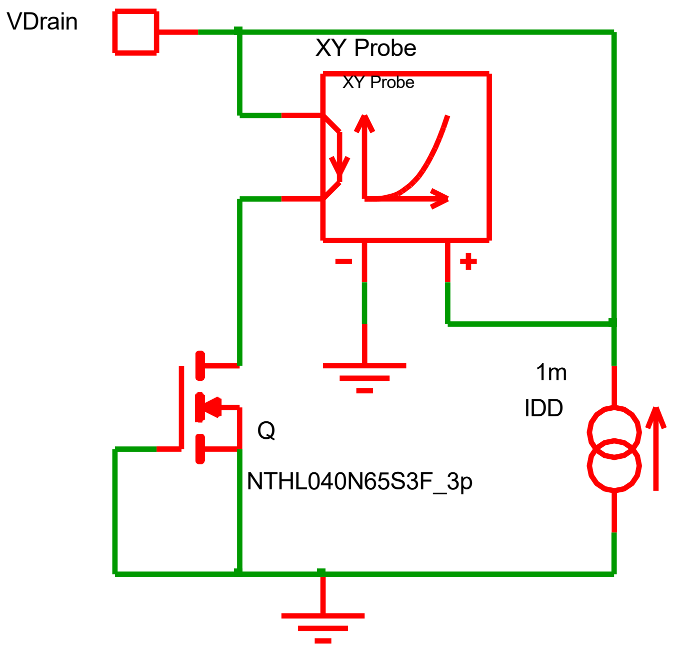
Simulation beyond the limits

Breakdown Voltage - Schematic

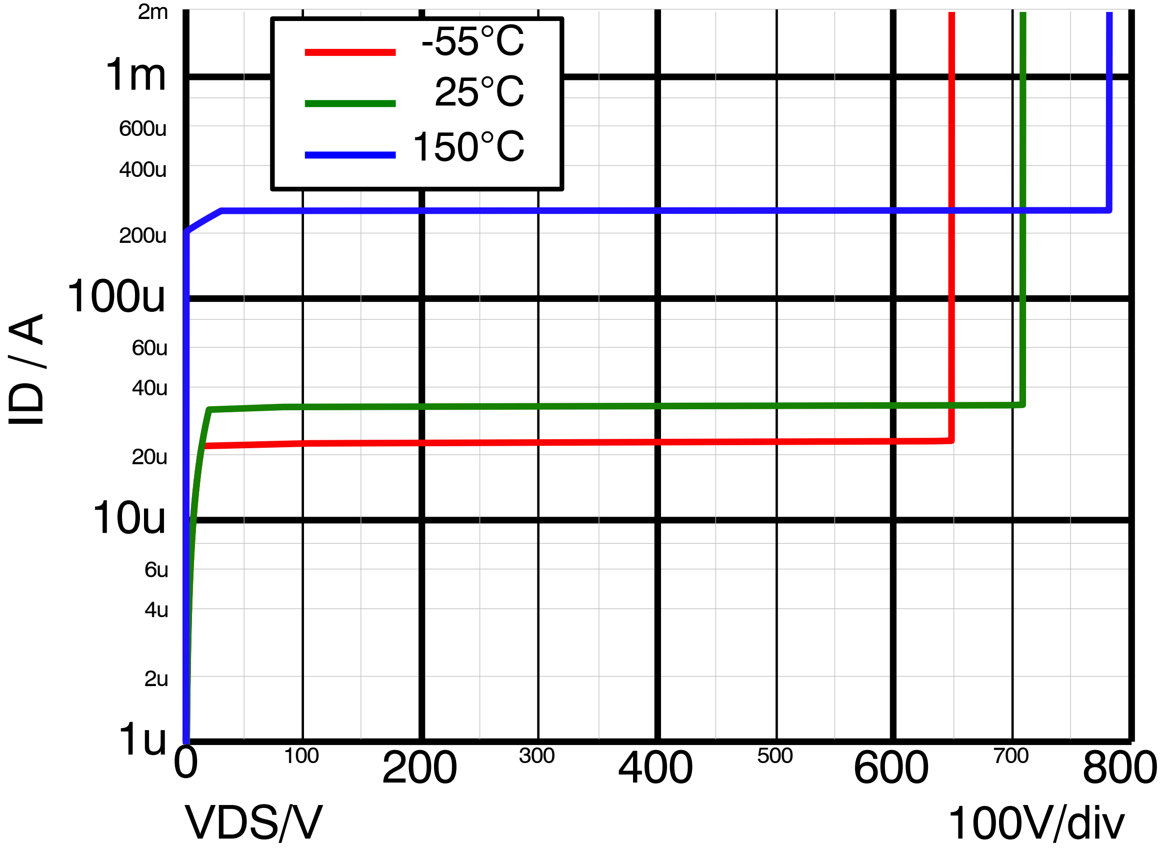
The model gives the average values. The model is accurate inside the specification limits. Results outside specification limits are not warranted.

But, the model can operate above the limits with relatively good accuracy and can predict values like the average breakdown drain to source voltage (BV_{DSS}).

To simulate the breakdown voltage depending on the temperature, we will use a ramp current source up to 2 mA and plot the “off” characteristic



Breakdown Voltage - Results



We can note the leakage current varies with temperature. We can measure 23 µA, 33 µA and 254 µA for respectively -55°C, 25°C and 150°C die temperature at a drain to source voltage equal to 400 V. We see a big leakage current increase between 25°C and 150°C.

The drain to source breakdown voltage is equal to 648 V, 708 V and 781 V for respectively -55°C, 25°C and 150°C die temperature.

Device Simulation Conclusion

So…

Simulation is a much safer environment for testing the limits and above…

All results in the data sheet can be obtained with those models.

As testing conditions in the data sheet are ideal (or not realistic),

- Simulation can provide more realistic characteristics,
- Parameters or Values in real conditions can be obtained.
- Parameters not in the data sheet can also be obtained.

Simulation models contain much more information than the data sheet.

DC-DC Boost Example

Evaluate losses and junction temperature

Application - Description

We will simulate a boost stage (for a solar inverter).

To avoid long stabilization time, we will close the loop. We will use a type 3 compensator with a voltage-mode pulse width modulator.

We will use arbitrary functions to calculate losses in the diode, in the MOSFET and the power to drive the MOSFET.

The “Per Cycle” measurement will be used to get the average losses for each cycle.

The specification for the power stage is the following:

Input voltage: 300 V,

Output voltage: 420 V,

Inductor current: 4 A,

Inductor current ripple: 2 A,

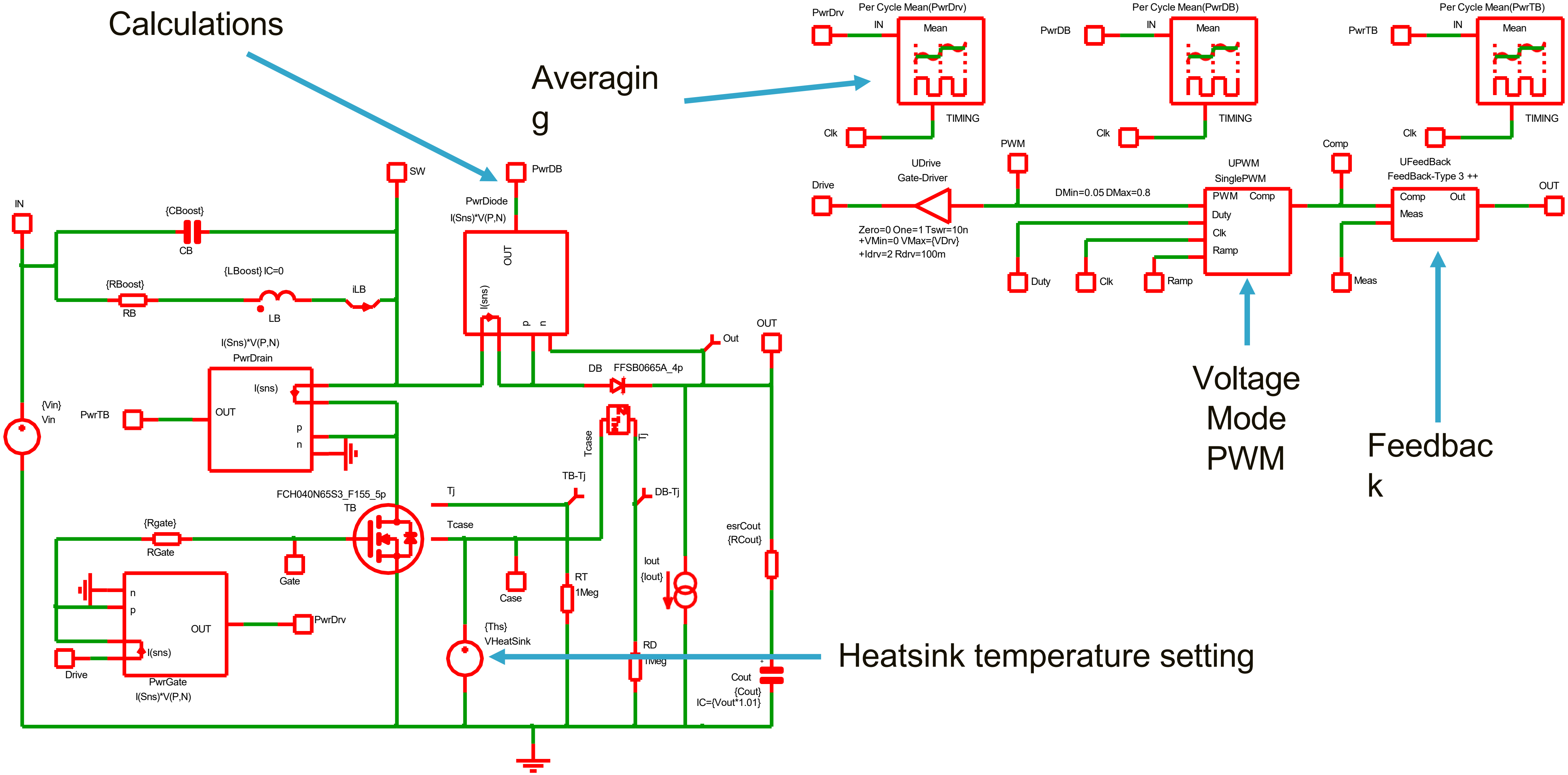
Switching frequency: 100 kHz,

Case temperature: 90 °C,

Gate drive voltage: 10 V,

Gate series resistor: 8 ν.

Application - Schematic



Losses Calculations

Averaging

Voltage Mode PWM

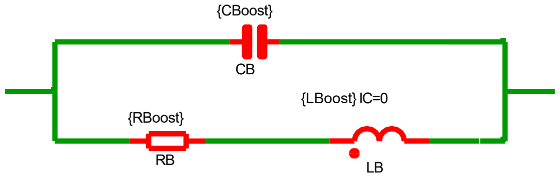
Feedback

Heatsink temperature setting

Application - Waveforms

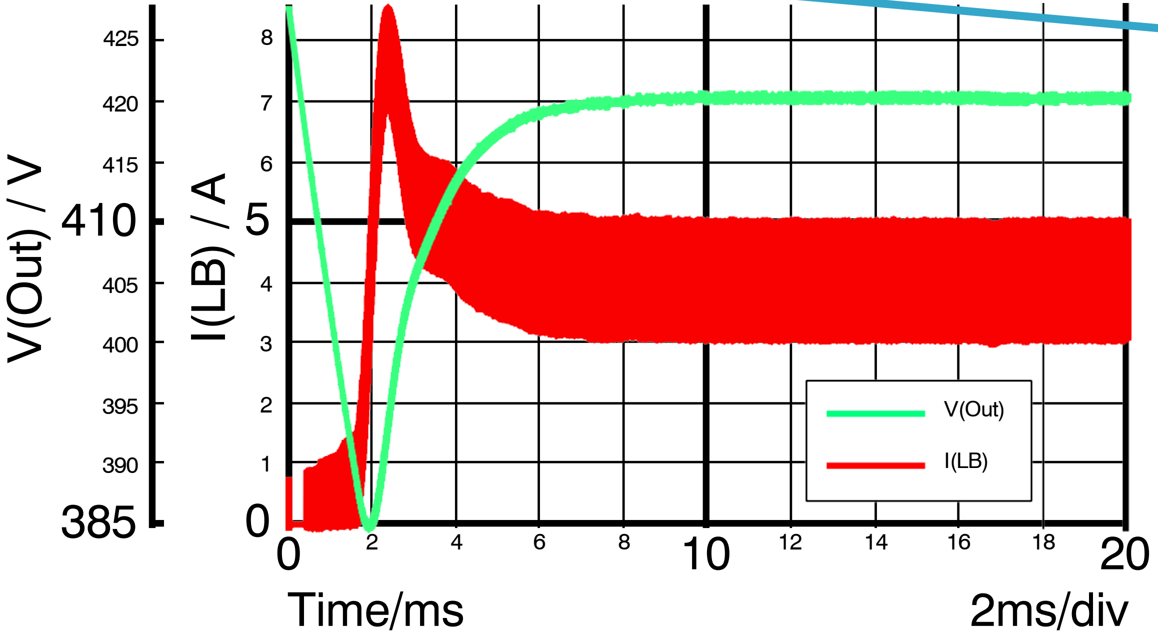
Ideal Inductor = LB only

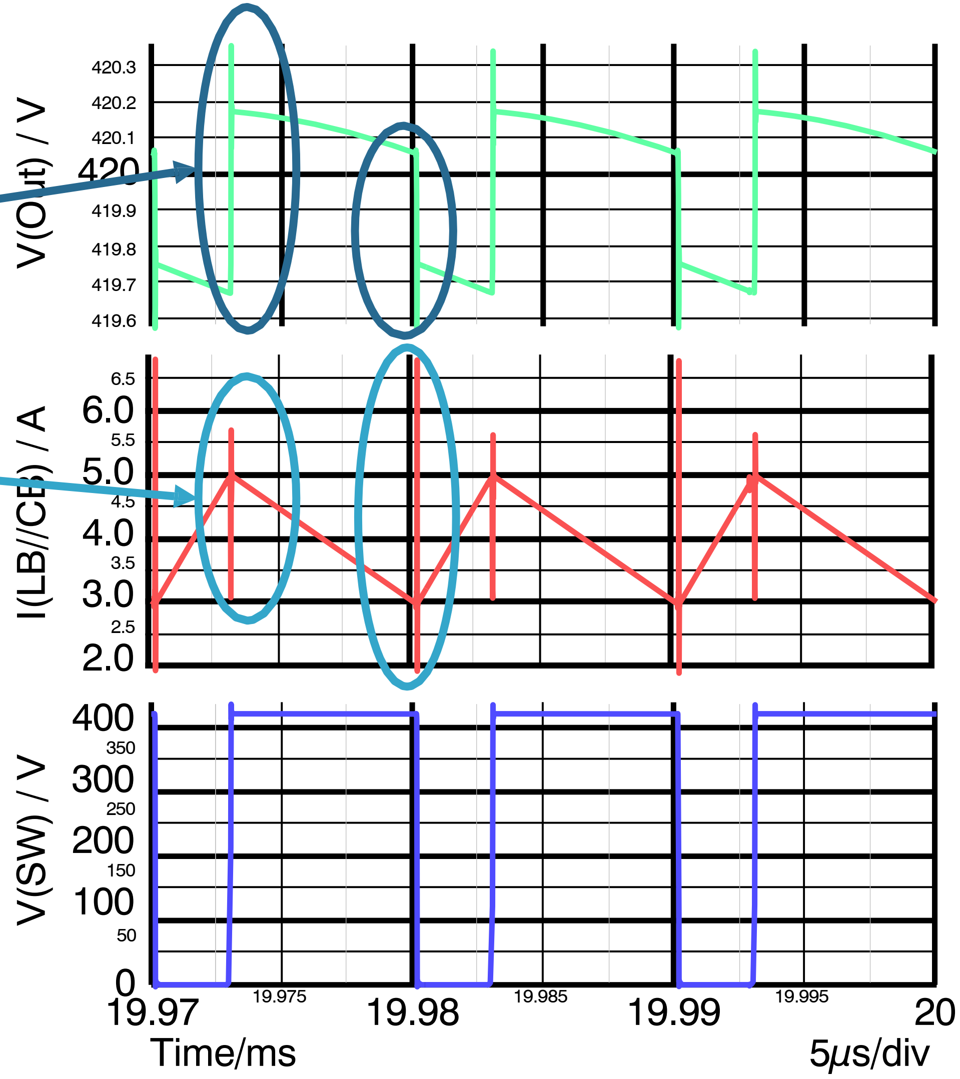
Real inductor = (LB+RB)//CB (with parasitics)



CB generate spikes on:

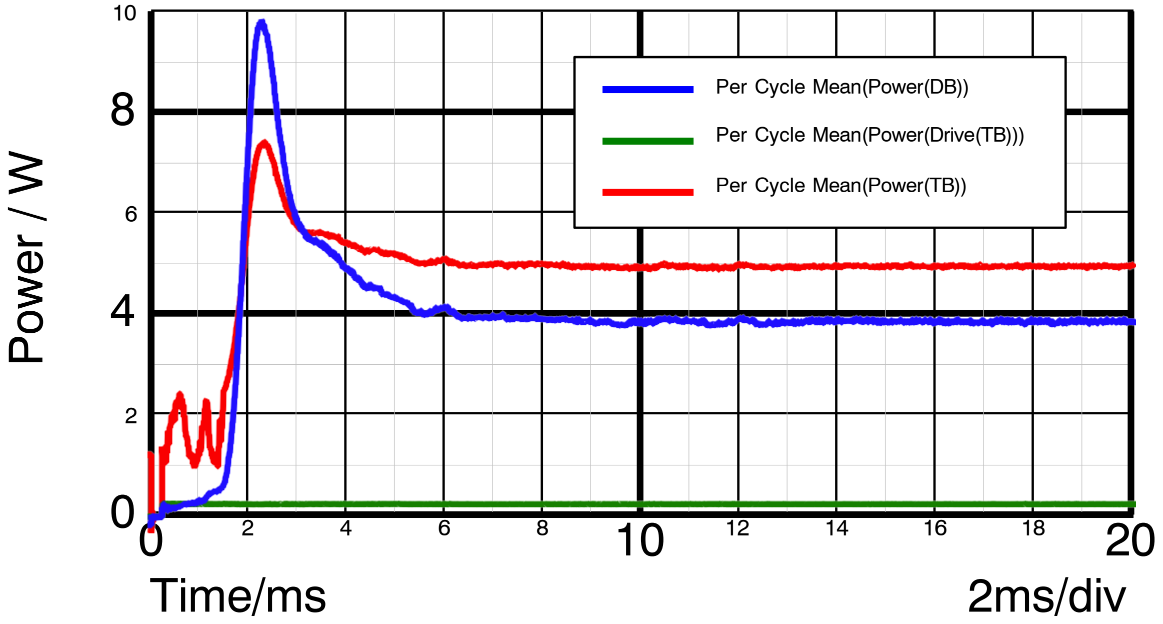
1) Output voltage
2) Input current

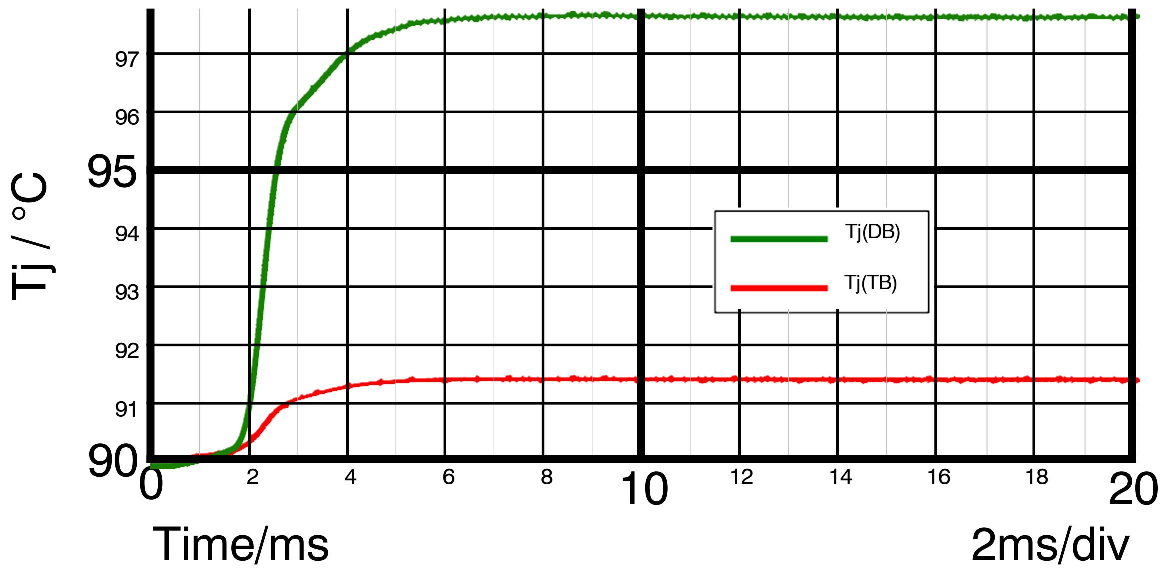




Application – Losses & Junction Temperature

For CB = 100 pF



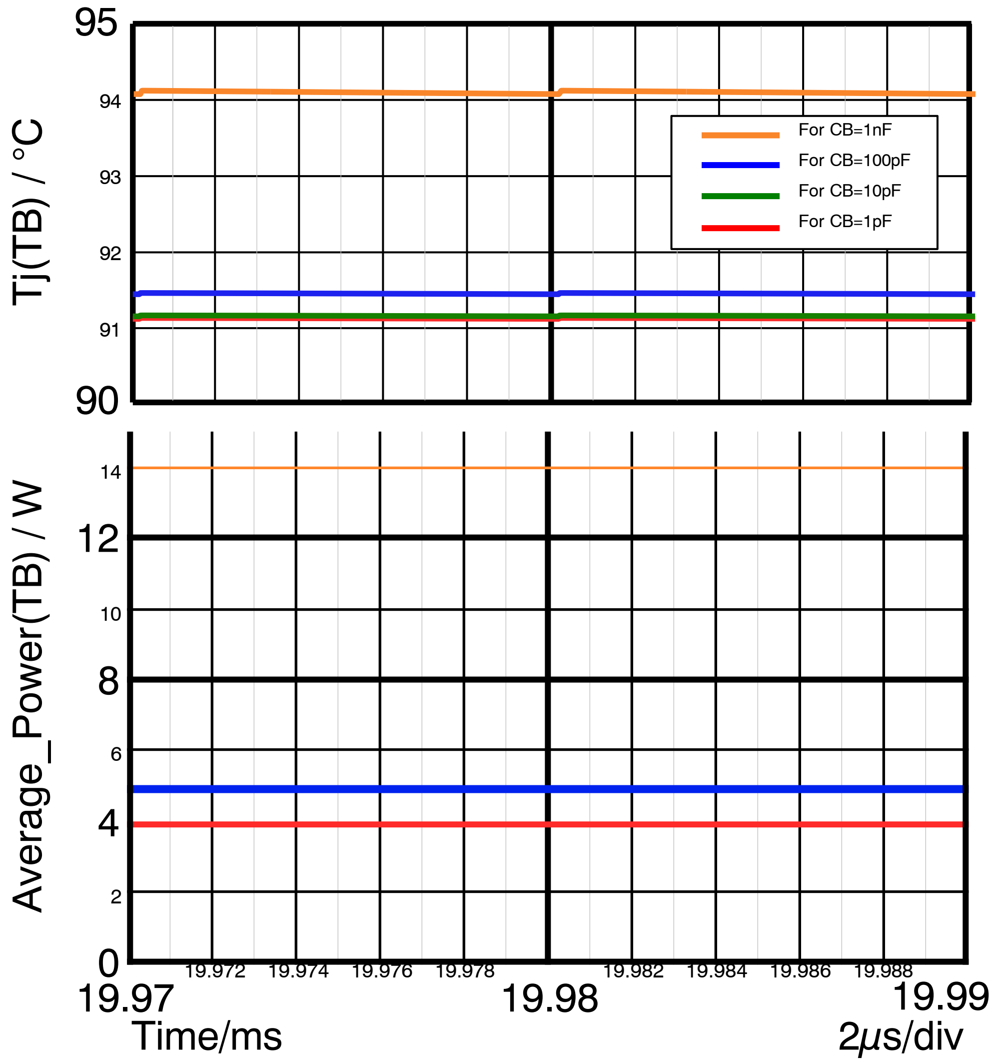


Application – Losses & Junction Temperature vs CB

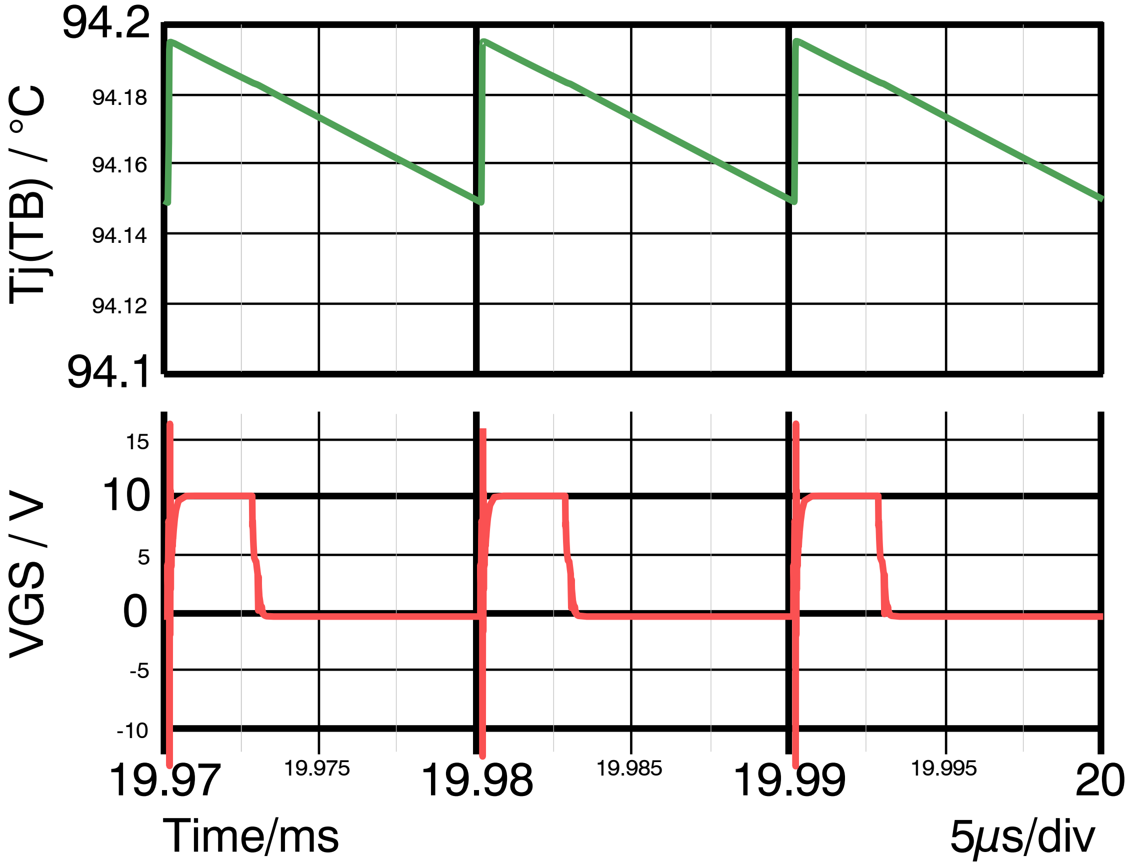
The diode losses and the driving losses are not affected by CB values.

We can measure the following values for the transistor as a function of the parasitic capacitor:

CB=1 pF => 3.9 W losses & $\phi\lambda(T_J)$=1.1°C
CB=10 pF => 4.0 W losses & $\phi\lambda(T_J)$= 1.2°C
CB=100 pF => 4.9 W losses
& $\phi\lambda(T_J)$= 1.4°C
CB=1 nF => 14.0 W losses & $\phi\lambda(T_J)$= 4.1°C



Application – Junction Temperature analysis



There is a step in junction temperature waveform.

This step is synchronous with turn on.

So, there is a peak of power losses during turn on link to CB!

Performance can be jeopardized by your inductor (*<u>parasitic capacitor</u>*)!

Application – Optimization

We keep the 100-pF inductor parasitic capacitor because this is a realistic value.
The output power is equal to 1.2 kW. We obtain the following table after several trials using TO220 package for the diode and TO247 for the MOSFET.

MOSFET (TB)	Diode (DB)	τι Tj MOSFET	τι TJ Diode	Drive Losses	MOSFET Losses	Diode Losses	Total Losses
FCH040N65S3	FFSP0465A	1,4 °C	7,3 °C	0,26 W	4,75 W	4,44 W	9,45 W
FCH040N65S3	FFSP0665A	1,5 °C	10,0 °C	0,26 W	4,98 W	3,88 W	9,12 W
FCH040N65S3	FFSP0865A	1,5 °C	4,1 °C	0,26 W	5,26 W	3,63 W	9,15 W
FCH040N65S3	FFSP1065A	1,6 °C	3,9 °C	0,26 W	5,46 W	3,41 W	9,13 W
FCH040N65S3	FFSP1265A	1,7 °C	3,7 °C	0,26 W	5,67 W	3,28 W	9,21 W
FCH040N65S3	FFSP0665B	1,4 °C	13,5 °C	0,26 W	4,80 W	3,80 W	8,86 W
FCH040N65S3	FFSP1065B	1,5 °C	6,6 °C	0,26 W	5,09 W	3,30 W	8,65 W
FCH067N65S3	FFSP0665B	1,6 °C	13,6 °C	0,15 W	3,90 W	3,80 W	7,85 W
FCH099N65S3	FFSP0665B	1,3 °C	13,6 °C	0,11 W	3,79 W	3,78 W	7,68 W
FCH125N65S3R0	FFSP0665B	1,7 °C	13,6 °C	0,09 W	3,85 W	3,79 W	7,73 W
FCH099N65S3	FFSP1065B	1,5 °C	6,6 °C	0,11 W	4,16 W	3,30 W	7,57 W

The last configuration with 10 A SiC new generation diode (FFSP1065B) and 99 mο EasyDrive SuperFET 3 (FCH099N65S3) gives less losses…
But a 10 A SiC diode for an average output current equal to 2.8 A could be consider oversized and, so, expensive.

Application Simulation Conclusion

So…

- Simulation gives a more realistic losses values than an estimated analytic function.
- Simulation can help to understand parasitic influences on an application schematic
- Running several configurations can help to find a better compromise between performances and cost.

Conclusion

Conclusion

Physical and scalable models can really help designers to analyze components characteristics and applications performances.

onsemi™

Intelligent Technology. Better Future.

Follow Us @onsemi

www.onsemi.com

Bibliography

- [1] “Physically Based, Scalable SPICE Modeling Methodologies for Modern Power Electronic Devices” https://www.onsemi.com/pub/Collateral/TND6260-D.PDF
- [2] “SPICE Modeling Tutorial”, https://www.onsemi.com/pub/Collateral/TND6248-D.PPTX
- [3] “How to use Physical and Scalable Models with SIMetrix, OrCAD and LTSpice” https://www.onsemi.com/pub/Collateral/AND9783-D.PDF
- [4] “Using Physical and Scalable Simulation Models to Evaluate Parameters and Application Results” https://www.onsemi.com/pub/Collateral/TND6330-D.PDF
- [5] “Simulate with Physical and Scalable Discrete Models…What could we get ?” https://www.onsemi.com/pub/Collateral/TND6329-D.PDF