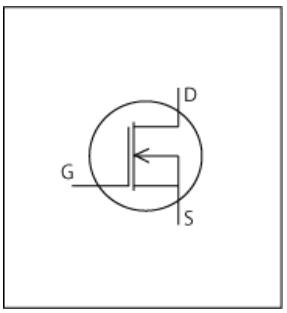
PSpice Model
NMOS
Renesas Electronics Corporation
2SK1516



Model Information

Model	A macro model based on BSIM3 model	
Call Name	MDC_2SK1516_PS	
Pin Assign	1:G 2:D 3:S	
File List	Model Library	MDC_2SK1516_PS01.lib
	Model Report	MDC_2SK1516_PS.pdf (this file)
Verified Simulator Version		PSpice version 17.2
Note		

References

The information which was used for modeling is as follow:

[Data Sheet]

- ●Date/Version Unknown
- ●Product name 2SK1516
- ●Company name Renesas Electronics Corporation
- ●Characteristics IdVgs[Temp],IdVds[Vgs],VdsVgs[Id],Rds(on)Id[Vgs],Rds(on)Temp[Id],Crss,Ciss,Coss,VgsQg[Vdd],VdsQg[Vdd],IsVsd[Vgs],tdon,tdoff,tf,tr

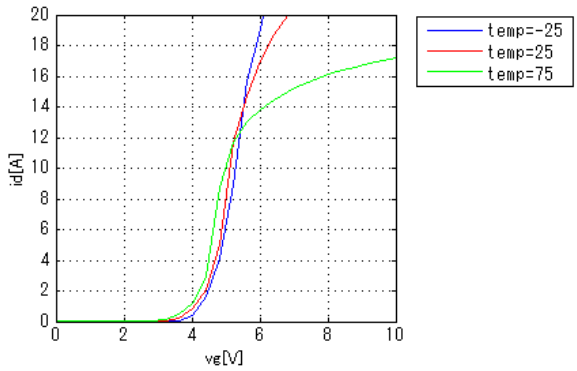
Simulation Range

This table shows the range of evaluated simulation range that was not occurs any convergence problems in this area.

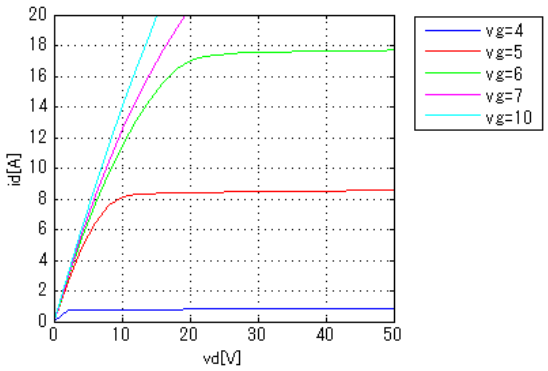
Item	Range			Unit
	Min.		Max.	
Drain-source voltage (DC)	0	to	500	V
Gate-source voltage (DC)	0	to	30	V
Temperature	-55	to	150	deg C

Simulation results are following.
Explanatory notes — : simulated

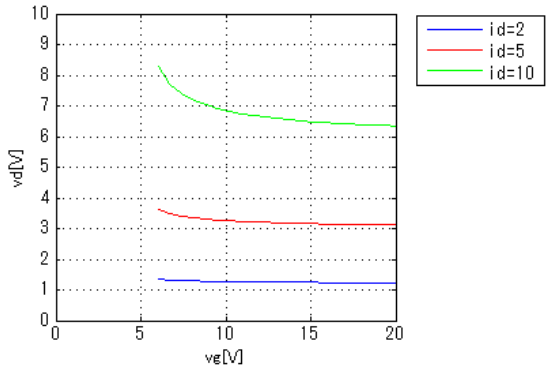
IdVgs[Temp]
Vds = 20V



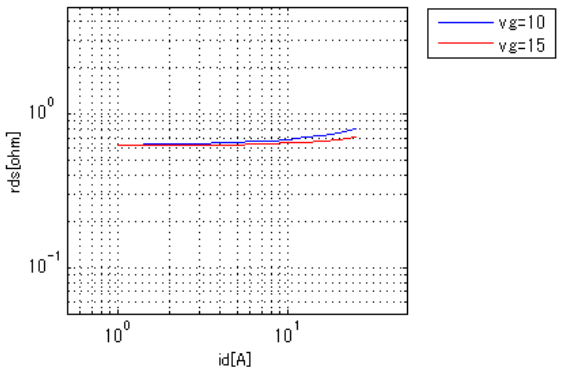
IdVds[Vgs]
Temp. = 25deg C



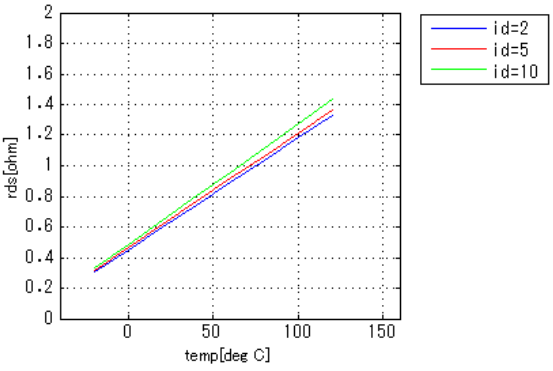
VdsVgs[Id]



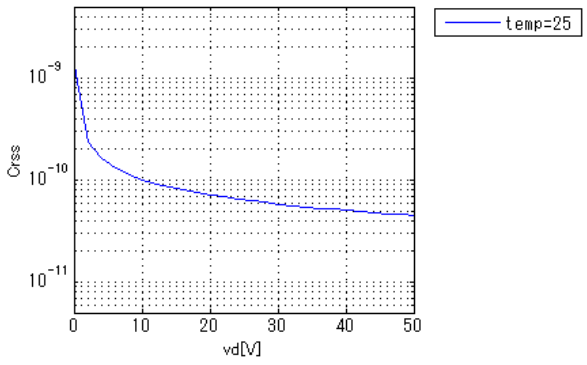
Rds(on)Id[Vgs]



Rds(on)Temp[Id]
Vgs = 10V

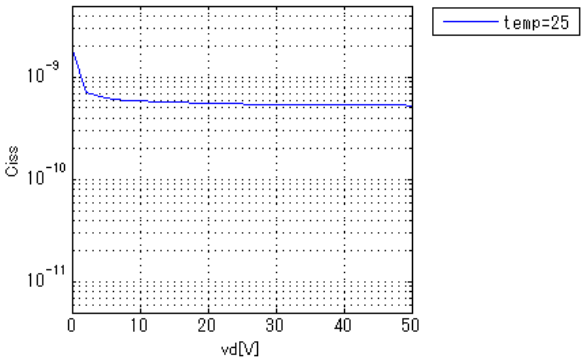


Crss
Freq. = 1MHz

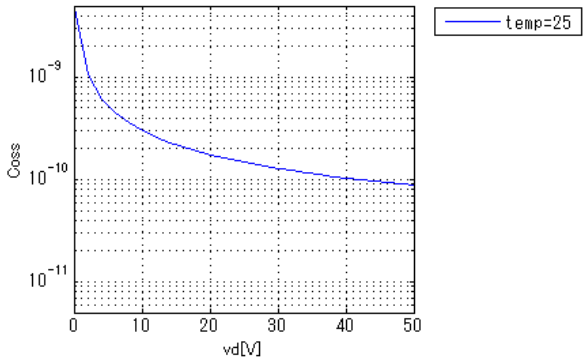


Simulation results are following.
Explanatory notes — : simulated

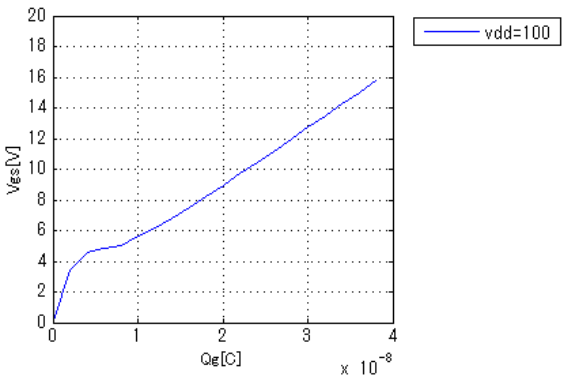
Ciss
Freq. = 1MHz



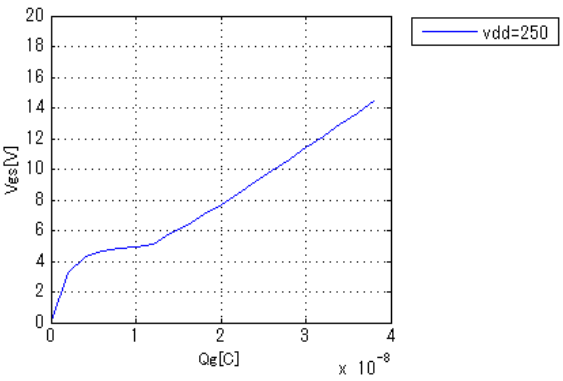
Coss
Freq. = 1MHz



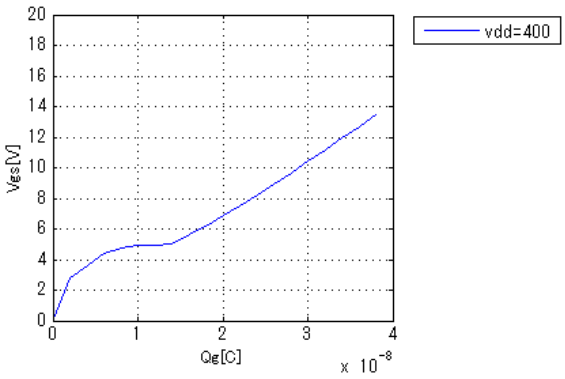
VgsQg[Vdd]
Id = 7A



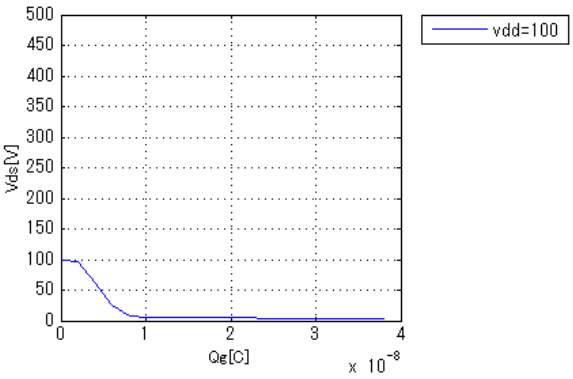
VgsQg[Vdd]
Id = 7A



VgsQg[Vdd]
Id = 7A

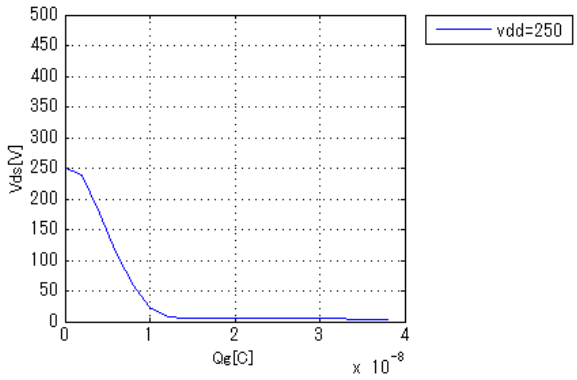


VdsQg[Vdd]
Id = 7A

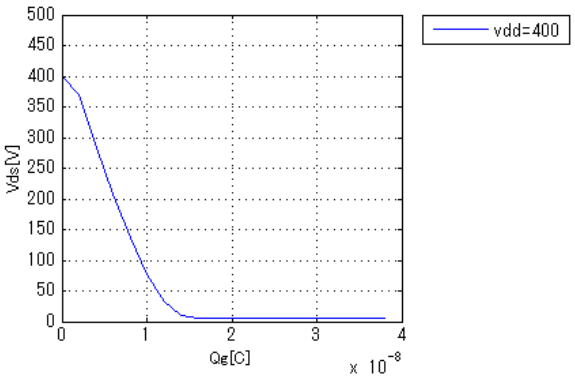


Simulation results are following.
Explanatory notes — : simulated

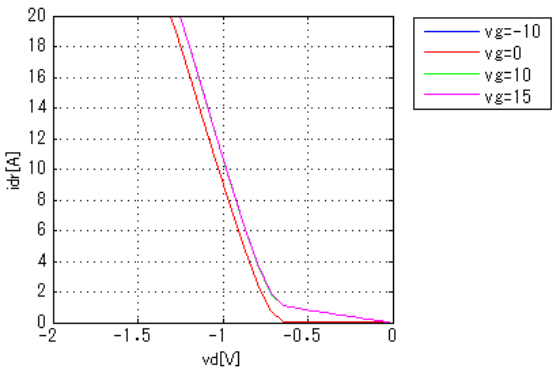
VdsQg[Vdd]
Id = 7A



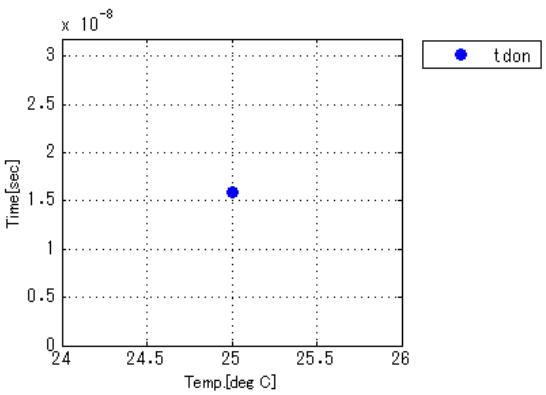
VdsQg[Vdd]
Id = 7A



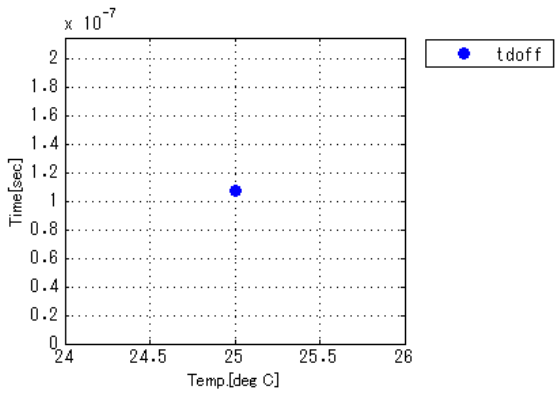
IsVsd[Vgs]



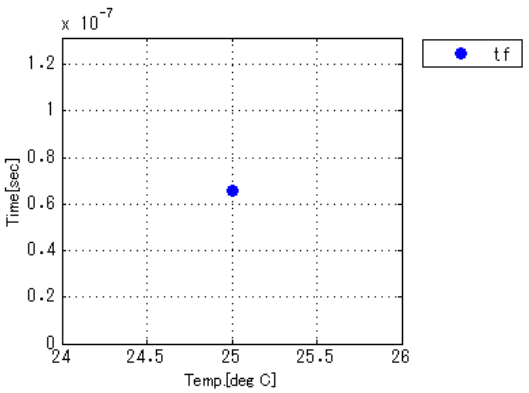
tdon
Vdd = 30V, Id = 5A, +Vg = 10V, -Vg = 0V, Rg = 0.001ohm



tdoff
Vdd = 30V, Id = 5A, +Vg = 10V, -Vg = 0V, Rg = 0.001ohm



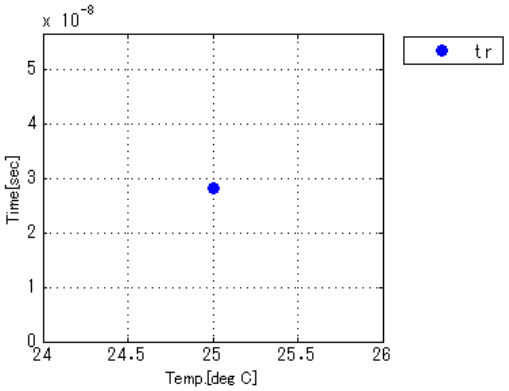
tf
Vdd = 30V, Id = 5A, +Vg = 10V, -Vg = 0V, Rg = 0.001ohm



Simulation results are following.
Explanatory notes — : simulated

tr

Vdd = 30V, Id = 5A, +Vg = 10V, -Vg = 0V, Rg = 0.001ohm



DISCLAIMER

1. This SPICE (Simulation Program with Integrated Circuit Emphasis) model and its content (the “Contents”) are copyright of MoDeCH Inc. All rights reserved. Any redistribution or reproduction of any or all part of the Contents in any form is prohibited without express written permission made by MoDeCH Inc.
2. MoDeCH Inc. as licensor (the” Licensor”) hereby grants to you, as licensee (the “Licensee”), a non-exclusive, non-transferable license to use the Contents as long as you abide by the terms and conditions of this DISCLAIMER.
3. The Licensee is not authorized to sell, loan, rent and redistribute or license the Contents in whole or in part, or in modified form, to anyone.
4. The Licensor shall in no way be liable to the Licensee or any third party for any loss or damage (including ,but not limited to, lost profits, or other incidental, consequential, or punitive damages), however caused (including through negligence) which may be directly or indirectly suffered from, arising out of, or in connection with, any use of the Contents .
5. Notwithstanding anything contained in this DISCLAIMER, in no event shall Licensor be liable for any claims, damages or loss which may arise from the modification, combination, operation or use of the Contents with the Licensee’s computer programs.
6. The Licensor does not warrant that the Contents will function in any environment.
7. The Contents may be changed or updated without notice. MoDeCH Inc. may also make improvements and/or changes in the products, pricing and/or the programs related to the Contents at any time without notice.

Modeling Design Technology

MoDeCH Inc.

Head Office
Location: 5-15 Yokoyama-cho, Hachioji-Shi, Tokyo 192-0081, Japan
Tel:+81-42-656-3360
E-Mail:model-on-support@modech.co.jp
URL:http://www.modech.com/en/