

A Power MOSFET SPICE Model with Built-In Model Generator

By Andre Adrian

Changing the SPICE model is now as easy as changing the schematic.

Introduction

The electronic circuit simulator SPICE has built-in MOSFET models. These models are based on lateral MOSFETs with a bulk connection. A power MOSFET has a vertical structure without bulk connection. This article presents the power function power MOSFET (PFPM) SPICE model.

The novelties in PFPM are the built-in model generators. The user provides the PFPM model generators with some coordinate points from the transfer characteristics, output characteristics, capacitances characteristics and the body diode forward characteristics. We can get the characteristics points from the data sheet or from measurements [1], [2]. At the beginning of every simulation run, these characteristics points are used to calculate power function coefficients. Changing the SPICE model is now as easy as changing the schematic.

Power MOSFET Equivalent Circuit

Figure 1 shows the power MOSFET equivalent circuit [3]. It has a voltage-dependent current source G to simulate the transconductance g_{fs} and the drain-source on-state resistance R_{dson} , a body-drain diode D and three parasitic capacitances C_{gs} , C_{ds} and C_{gd} . The bond wire inductances are L_s , L_d and L_g . R_g models the distributed resistance of the gate structure.

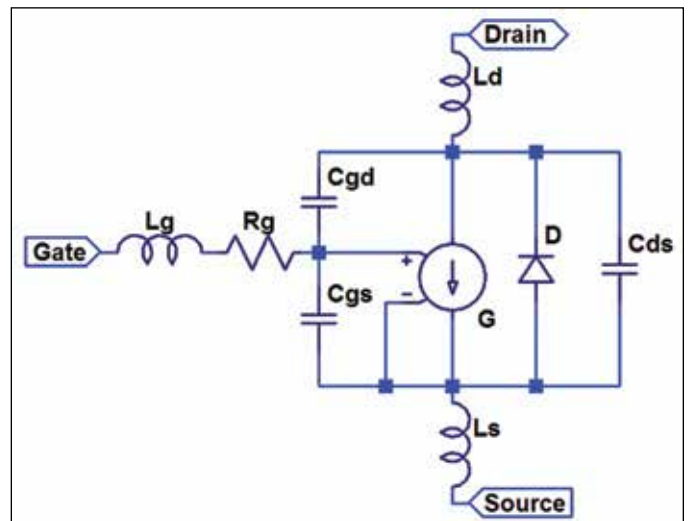


Figure 1 • Power MOSFET equivalent circuit.

Model Verification

For model verification we use a Hartley oscillator circuit. The Huth-Kühn variant of the Hartley circuit uses the C_{gd} capacity as one of the three feedback reactances [4]. The inductances L_1 , L_2 complete the feedback network. The parasitic capacitances C_{gs} and C_{ds} are parallel to the inductances and influence the oscillator frequency. If the SPICE simulation and the lab buildup agree in resonance frequency and in wave form, we can assume that the model reflects reality. The PFPM is able to simulate the MOSFET behavior for HF and VHF frequencies in small-signal and large-signal applications. But the model is not complete. Temperature effects are not modeled.

The inductance L_{stray3} is created by the conductor path between source and ground. A straight wire of 0.6mm diameter has an inductance of 7nH per centimeter. The circuit oscillates at 21MHz. If there are only stray inductances in the circuit, oscillation frequency is 77MHz. The unity gain frequency f_T is not given in a MOSFET datasheet. But we can add up the times for switching t_{don} , t_r , t_{doff} and t_f to calculate the time for one period of a triangle wave that swings

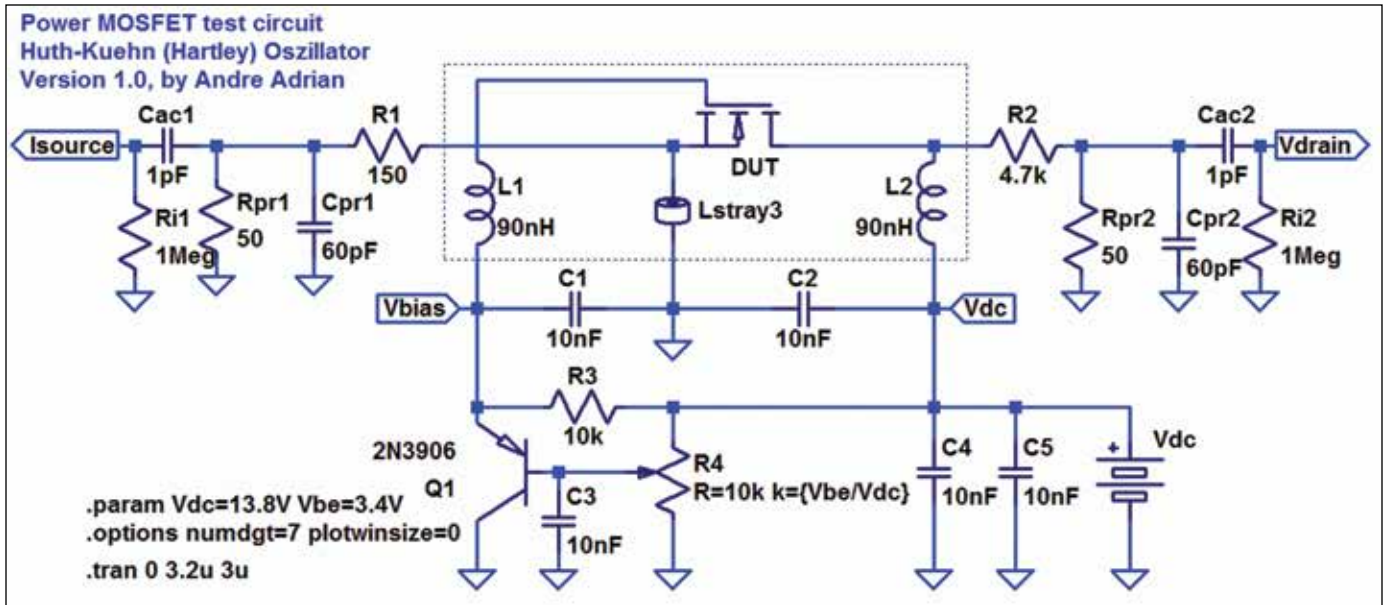


Figure 2 • Huth-Kühn (Hartley) oscillator. All frequency determining components are in the dashed lines box. The load R1, R2, ... will decrease the oscillator frequency a bit.



Picture 1 • Breadboard implementation of Huth-Kühn oscillator.

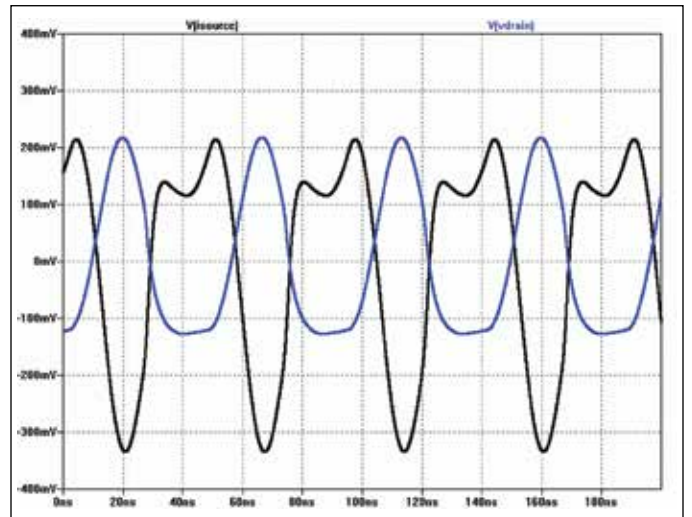


Figure 3 • SPICE simulation of Isource (black) and Vdrain (blue).

between 10% and 90% of the drain supply voltage. The Infineon MOSFET IPP530N15N3 [5] has a typical period length of 34ns or a “triangle frequency” of 29.4MHz.

The components left of resistor R1 and right of R2 simulate the oscilloscope probe (Cpr, Rpr) and the AC mode of the oscilloscope (Cac, Ri). The transistor Q1 provides a low impedance bias voltage for the MOSFET DUT (device under test). The capacitors C1 to C5 are blocking/filter capacities. Ceramic or mica capacitors with short leads or SMD components work best. The coils L1 and L2 have an inductance of 70nH. The additional 20nH are stray inductance. Lstray3 has 10nH. Fig. 3 shows the Isource and Vdrain waveforms in LTSpice simulation. Pic. 2 shows an oscilloscope screen shot. The Tektronix 7A26 amplifier in the 7834 chassis has 200MHz bandwidth.

The simulation waveform shows a saturation “dent” at the top of every Isource wave. The dent is less prominent in the oscilloscope picture. The bottom of the Vdrain curve in the oscilloscope has some ripple, which the simulation waveform has not.

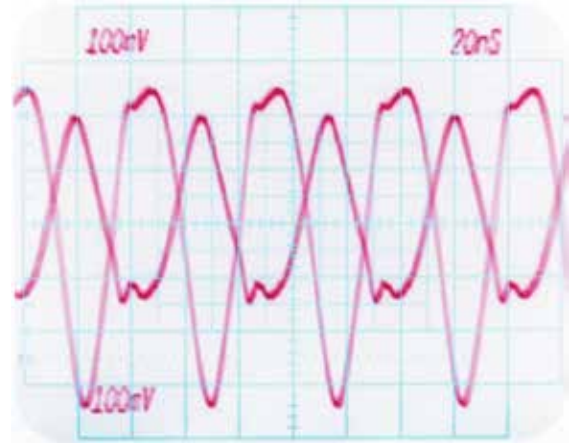
And now we look how the model works and how the model parameters are extracted from the datasheet.

The Body Diode Forward Current Function

The most simple transfer function for a MOSFET body diode is the piecewise linear model (PWL). The PWL function is monotonically increasing, but not strictly increasing. The PWL characteristic has a kink. In nature everything is continuous. This is true for any MOSFET curve, too.

The power function (PF) is a strictly monotonic function for all valid x values. The square function, the square root function and the reciprocal function are power functions. In a real diode, the slope is not constant, but changes with the forward current. The coefficients threshold voltage V_{to} , scaling factor c and exponent r control the PF. Using two points from the current–voltage characteristic, c and r can be calculated. This parameter extraction is without error, if the characteristic is a PF.

From the IPP530N15N3 MOSFET datasheet we extract V_{to} as 0.7V, point one as 0.75V; 1A and point two as 0.95V; 20A.



Picture 2 • Oscilloscope screen shot of I_{source} and V_{drain} .

$$V_{to} := 0.70$$

$$x_1 := 0.75 - V_{to} \quad y_1 := 1 \quad x_2 := 0.95 - V_{to} \quad y_2 := 20$$

The variables x_1 , x_2 are forward voltage V_f minus threshold voltage V_{to} . This substitution eliminates V_{to} . The power functions for the two points are:

$$y_1 = c \cdot x_1^r \quad y_2 = c \cdot x_2^r$$

First step is to resolve the left equation to c , then replace c in the second equation and resolve for r . With r and one point we can calculate c .

$$c = \frac{y_1}{x_1^r} \quad y_2 = y_1 \cdot \frac{x_2^r}{x_1^r} \quad r = \frac{\ln(y_2) - \ln(y_1)}{\ln(x_2) - \ln(x_1)}$$

For the IPP530N15N3 body diode the PF parameters are:

$$r=1.861 \quad c=264.044$$

The power function for I_f is:

$$I_f(x) := c \cdot (x - V_{to})^r$$

Figure 4 shows the PWL model (red line) and the PF model (blue line). The blue line should agree with the body diode forward characteristics in the data sheet.

The MOSFET Transfer Function

The MOSFET transfer characteristics and the body diode forward characteristics have the same shape. The difference is in the threshold voltage. The MOSFET gate needs a threshold voltage V_{to} of 3 to 5 volts to allow some drain current to flow.

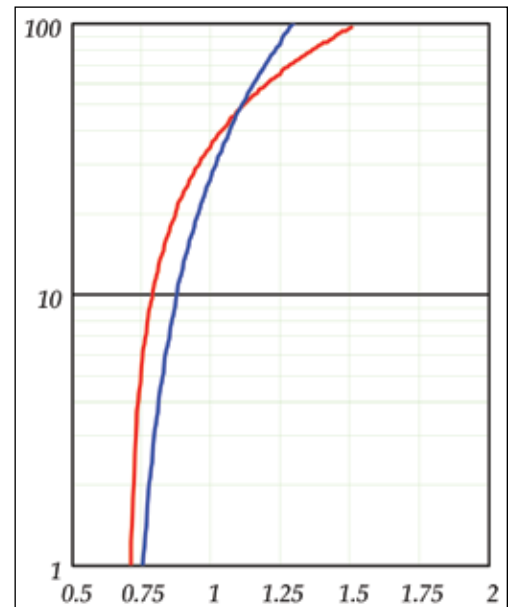


Figure 4 • Diode Transfer function: x axis shows forward voltage, y axis shows forward current.

In the transfer characteristics we find point one as 4.88V; 5A and point two as 5.59V; 15A. The model generator parameters are:

$$V_{to} := 3.8$$

$$x_1 := 4.88 - V_{to} \quad y_1 := 5 \quad x_2 := 5.59 - V_{to} \quad y_2 := 15$$

The power function coefficients for the transfer function are:

$$r=2.174 \quad c=4.23$$

The MOSFET Output Function

The MOSFET output function describes the amplification. The output characteristics show that there are two independent variables, Vds and Vgs and one dependent variable Id. The drain connector of the MOSFET is used for input and output. For every time step, SPICE uses first the drain as input to get the actual drain/source voltage (Vds) and second to drain the drain current (Id) or the body diode current.

The Output Function Regions

The output function has three regions in the Shichman-Hodges MOSFET model: the cutoff region, the ohmic or linear region and the saturated region. The SPICE NMOS model sets in the cutoff region the drain current to zero. We do the same.

The PFPM model uses different functions for ohmic and saturated region drain current and the min function to combine them. The min function creates a characteristic with a kink. Averaging is the feat to smooth the kink. At every time step the drain currents for three different gate voltages are calculated. The simulation uses the average current for further processing.

The ohmic region output function uses the square root of Vds divided by the Rdson resistance to calculate Id. The output function for the linear region is the MOSFET transfer function.

The output function Id() uses gate/source voltage Vgs and drain/source voltage Vds as input variables. Output variable is the drain current Id.

$$R_{dson} := 44 \cdot 10^{-3} \quad Id(V_{gs}, V_{ds}) := \min \left[\frac{\sqrt{V_{ds}}}{R_{dson}}, c \cdot (V_{gs} - V_{to})^r \right]$$

The Idavg() function performs the smoothing. The parameter d is the distance in volt between the Vgs values.

$$d := 0.1 \quad Idavg(V_{gs}, V_{ds}) := \frac{(Id(V_{gs}, V_{ds}) + Id(V_{gs} + d, V_{ds}) + Id(V_{gs} - d, V_{ds}))}{3}$$

Parasitic Capacities

The parasitic capacities in a power MOSFET are large and voltage dependent. Together with the bond wires inductances or with other (stray) reactances they can form LC circuits which cause ringing (very high frequency oscillation). The power MOSFET has three parasitic capacities. Cds is the capacity of the body (reverse) diode. Every diode is a varactor (capacity diode). Cgs and Cgd are capacities between gate and the channel.

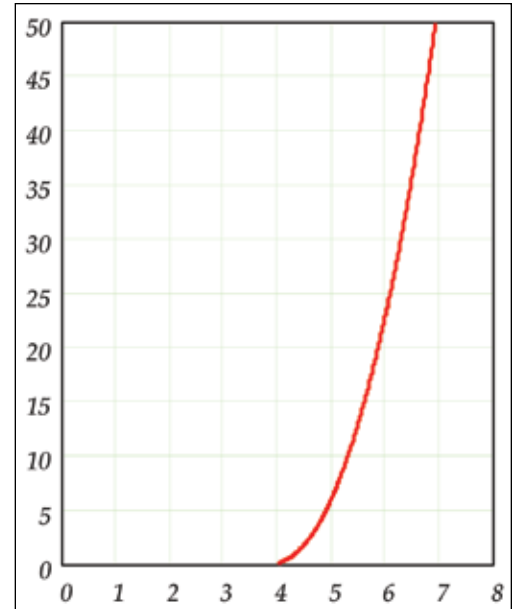


Figure 5 • MOSFET Transfer function: x axis shows gate to source voltage; y axis shows drain current.

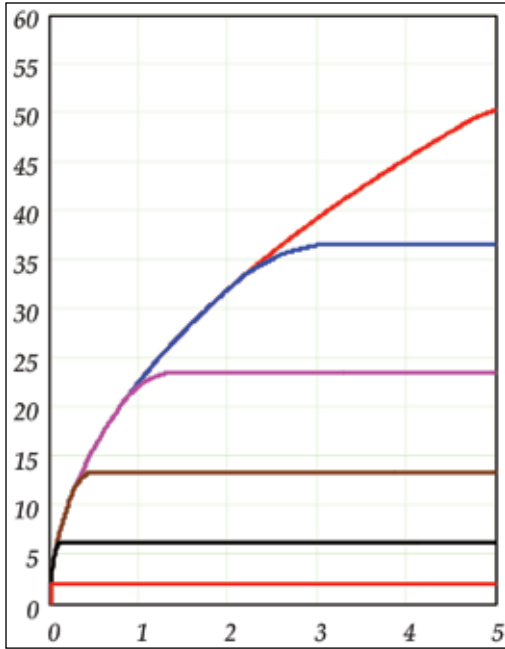


Figure 6 • Output function: x axis shows Vds in volt; y axis shows Id in ampere. Parameter lines are Vgs in volt. Values from bottom to top are 4.5V to 7V in 0.5V steps.

A power function is used to model the voltage dependent capacities Cgd and Cds. With independent variable x for the voltage across the capacity and the dependent variable C for the resulting capacity the formula is:

$$C = C_0 \cdot (1 + c \cdot x)^r$$

C0 is the varactor capacity at zero volts (Cgd0 or Cds0). The parameters c and r are again the power function coefficients. We introduce $y = C/C_0$ to eliminate C0:

$$y = \frac{C}{C_0} \quad y = (1 + c \cdot x)^r$$

Recursive Model Generator Formula

For the parameter estimation we need two points from the capacity curve to calculate c and r. The author and Mathcad do not know a closed-form solution for the model generator. We need a recursive (iterative) formula. The Ansatz (beginning) is the same as above. Isolate a coefficient and plug it into the second formula. First we calculate the Cgd power function coefficients:

In a datasheet we find the capacities Ciss, Coss and Crss. The relations between these parameters and the MOSFET model parameters are simple:

$$C_{gd} = C_{rss} \quad C_{gs} = C_{iss} - C_{rss} \quad C_{ds} = C_{oss} - C_{rss}$$

For the power functions we need the zero volts capacities and two more capacities. In the IPP530N15N3 MOSFET datasheet we find:

$$V_1 := 5 \quad V_2 := 40$$

$$C_{iss0} := 778 \cdot 10^{-12} \quad C_{iss1} := 707 \cdot 10^{-12} \quad C_{iss2} := 707 \cdot 10^{-12}$$

$$C_{oss0} := 1091 \cdot 10^{-12} \quad C_{oss1} := 661 \cdot 10^{-12} \quad C_{oss2} := 366 \cdot 10^{-12}$$

$$C_{rss0} := 90 \cdot 10^{-12} \quad C_{rss1} := 33 \cdot 10^{-12} \quad C_{rss2} := 4 \cdot 10^{-12}$$

The Cgs capacity is not very voltage dependent and is modeled as a constant capacity with:

$$C_{gs} := C_{iss1} - C_{rss1} \quad C_{gs} = 674 \times 10^{-12}$$

$$C_0 := C_{rss0}$$

$$x_1 := V_1 \quad y_1 := \frac{C_{rss1}}{C_0} \quad y_1 = 0.367$$

$$x_2 := V_2 \quad y_2 := \frac{C_{rss2}}{C_0} \quad y_2 = 0.044$$

$$y_1 = (1 + c \cdot x_1)^r \quad y_2 = (1 + c \cdot x_2)^r$$

The first formula is resolved for c. We replace c in the second formula and resolve for r. We see r at the left and at the right of the equal sign:

$$c = \frac{\left(\exp\left(\frac{\ln(y_1)}{r}\right) - 1 \right)}{x_1} \quad y_2 = \left[1 + \frac{x_2}{x_1} \cdot \left(\exp\left(\frac{\ln(y_1)}{r}\right) - 1 \right) \right]^r$$

$$f(r) := \frac{\ln(y_2)}{\ln\left[1 + \frac{x_2}{x_1} \cdot \left(\exp\left(\frac{\ln(y_1)}{r}\right) - 1 \right) \right]}$$

To calculate r, we start with the value -1 for r and use the recursive formula until convergence. We recognize convergence if the difference between the last two r values is below a threshold.

$$r := f(f(f(f(f(f(-1)))))) \quad r = -1.366$$

The limit of our function value is:

$$r_{limit} = -1.379$$

After six iterations we are below an error of 5%. The datasheet tells us that Coss at 75V Vds is typical 80pF and maximum 106pF. The differences between MOSFETs of the same type are larger than our modeling error. Because we need this recursion run only once, we can iterate ten times to get a better r value.

$$error := 1 - \frac{r}{r_{limit}} \quad error = 0.892 \%$$

Calculating c is easy.

$$c := \frac{\left(\exp\left(\frac{\ln(y_1)}{r}\right) - 1 \right)}{x_1} \quad c = 0.217$$

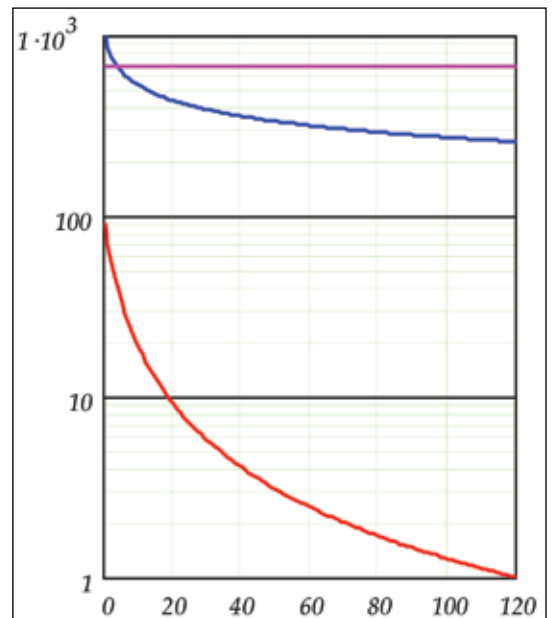


Figure 7 • Voltage dependent C characteristics: x axis shows Vds in volts; y axis shows capacity in Pico farad. Red curve is for Cgd, blue for Cds and magenta for Cgs.

The diagram Figure 7 shows the Cgd voltage to capacity curve in red. We see the typical hyperbolic (reciprocal) shape of a varactor curve.

Capacity to Charge Conversion

We cannot use the voltage to capacity function in SPICE, we need a voltage to charge (Q) function. The charge is the integral of the capacity.

$$C(x) = C_0(1 + c \cdot x)^r \quad Q = \int C_0(1 + c \cdot x)^r dx = \frac{C_0}{c \cdot (r + 1)} \cdot (1 + c \cdot x)^{(r+1)}$$

The final step is to combine the parameters into k, c and ρ:

$$\rho := r + 1 \quad k := \frac{C_0}{c \cdot \rho} \quad Q(x) := k \cdot (1 + c \cdot x)^\rho$$

The calculations for Cds are the same.

$$C_0 := C_{oss0} - C_{rss0}$$

$$y_1 := \frac{C_{oss1} - C_{rss1}}{C_0} \quad y_1 = 0.627 \quad y_2 := \frac{C_{oss2} - C_{rss2}}{C_0} \quad y_2 = 0.362$$

After six iterations with the start value -1 we get:

$$r = -0.303 \quad r_{limit} = -0.293 \quad error = -3.414\% \quad c = 0.732$$

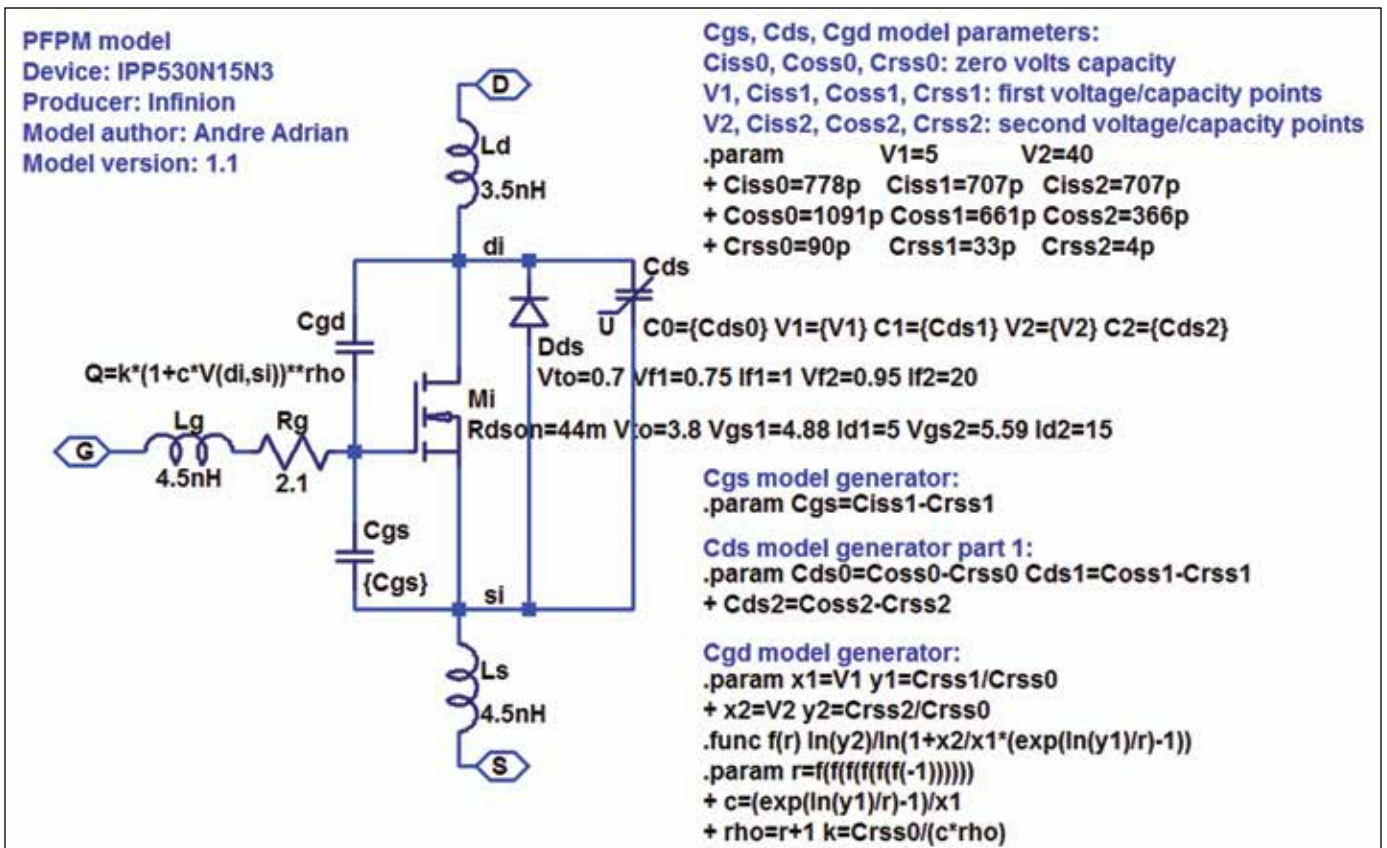


Figure 8 • Power MOSFET equivalent circuit with all parameters.

Usage of PFPM Model Generators

LTSpice is a well-known SPICE implementation. One nice LTSpice feature is that we can use the LTSpice schematics editor to implement the PFPM model generators and the PFPM model. Figure 8 shows the power MOSFET equivalent circuit again, but now with all details. Working with SPICE models was never easier.

The LTSpice model files can be found at <http://www.andreadrian.de/pfpm>.

About the Author:

Andre Adrian lives in Germany. After an apprenticeship at Siemens in Frankfurt he obtained a Computer Science degree from the Polytechnic (Fachhochschule) in Wiesbaden. He works for the German Air Traffic Control agency in the area of mission-critical, redundant, real-time computer systems.

References

- [1] International Rectifier; AN-957: Measuring HEXFET(R) MOSFET Characteristics
- [2] International Rectifier; AN-1001: A More Realistic Characterization of Power MOSFET Output Capacitance C_{oss}
- [3] International Rectifier, Vrej Barkhordarian; AN-1084: Power MOSFET Basics
- [4] Huth GmbH, Ludwig Kühn; German patent DE 310152, Schaltungsweise zur Schwingungserzeugung mit Vakuumröhren; applied 28.Sep.1917
- [5] Infineon; OptiMOS(TM) 3 Power-Transistor IPP530N15N3, Rev. 2.5; 2010-07-13