International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056
Volume: 04 Issue: 05 | May -2017 www.irjet.net p-ISSN: 2395-0072

VERIFICATION OF SPICE MODEL OF MOSFET

Vishal V. Bodake[1],Prof.Amutha Jeyakumar[2],Devendrabhai Patel[3]

[1]Student, Dept. of Electrical Engineering, VJTI, Mumbai, Maharashtra
[2]Associate Professor, Dept. of Electrical Engineering, VJTI, Mumbai, Maharashtra
[3]Technical specialist, Cummins India, Pune

---***---

Abstract - *Automotive industry uses many electronics controllers. These controllers contain discrete components like Resistor, Capacitor, MOSFET, Diode, Transistors etc. Electronic hardware development goes through design, analysis and hardware verification. Time and cost saving of controller development requires analysis of design in simulation environment before actual hardware build. Component simulation model plays key role in the accuracy of overall simulation. Inaccurate simulation model will give inaccurate results which may lead to over or under design. Hence accuracy of simulation model needs of be verified before using in simulation. Simulation model of component is widely available in SPICE format and this paper focuses on SPICE model. This paper explains process of verification of MOSFET SPICE model by using OrCAD.*

***Key Words*: SPICE, Simulation, OrCAD, MOSFET**

1. INTRODUCTION

SPICE i.e. Simulation Program with Integrated Circuit Emphasis is the circuit simulator program. SPICE model define the electrical behavior of part. Model definitions are stored in .lib file format. Simulation of PCB level circuits is the most common use of SPICE models. The PSpice OrCAD provides simulation environment for simulation of these SPICE models. These SPICE models of components are available on component manufacturer's website. In verification of these SPICE models we obtain the different characteristics of components and compare it with actual datasheet characteristics of that component. After getting verification result we find which SPICE model is accurate and we develop the library of accurate SPICE models of components. This verification not only gives accuracy result but also give functional behavior of components in the circuit. In this paper verification process is defined for MOSFET SPICE model for three different characteristics.

- Characteristics of MOSFET to be verify: Typical Output characteristics, Typical Transfer characteristics , Drain-Source On state resistance Vs Drain current

For Verification we use simulation tool PSpice. PSpice is a simulation program that models the behavior of a circuit containing any mix of analog and digital devices. Used with design entry tools such as OrCAD Capture or Design Entry HDL for design entry, you can think of PSpice as a software-based breadboard of your circuit that you can use to test and refine your design before even touching a piece of hardware.

To describe the circuit topology a netlist file is given as an input to the SPICE simulator. Netlist is the line statements that identify circuit elements, their electrical characteristics and their nodal connections.

For instance, the line **"R12 1 4 100 Kohm"** Defines a 100 Kohm resistor that is connected between nodes 1 and 4.

2. GENERAL VERIFICATION FLOW

A. Download SPICE model of component to be verify (.lib)

First download the SPICE model of component part to be verified from the manufacturer's website. Fig-1. is the example of SPICE model of MOSFET of Infineon whose manufacturing part number is IPD30N03S4L-14.

B. Create Symbol from model in PSpice tool Model editor (.olb)

Open Model editor and browse .lib file of model in it. Configure the pins of symbol with pins of SPICE model, then symbol is created with .olb extension as shown in Fig-2.

```
.SUBCKT IPD30N03S4L_14_L0 drain gate source
Lg    gate g1   3n
Ld    drain d1   1n
Ls    source s1  2n
Rs    s1  s2  2.35m
Rg    g1  g2   1.2
M1    d2  g2  s2  s2  DMOS  L=1u  W=1u
.MODEL DMOS NMOS ( KP= 77.9  VTO=2.62  THETA=0
VMAX=1.5e5  ETA=0.015  LEVEL=3)
Rd    d1  d2  7.16m TC=7m
Dbd   s2  d2  Dbt
.MODEL   Dbt   D(BV=38  M=0.3  CJO=0.96n  VJ=0.9V)
Dbody  s2  21  DBODY
.MODEL DBODY  D(IS=6.6p  N=1.1  RS=0.58u  EG=1.12  TT=3n)
Rdiode  d1  21   2.53m TC=10m
.MODEL  sw   NMOS(VTO=0  KP=10  LEVEL=1)
Maux    g2  c  a  a  sw
Maux2   b  d  g2  g2  sw
Eaux    c  a  d2  g2  1
Eaux2   d  g2  d2  g2  -1
Cox     b  d2  0.18n
.MODEL   DGD   D(M=0.6  CJO=0.18n  VJ=0.5)
Rpar    b  d2  1Meg
Dgd     a  d2  DGD
Rpar2   d2  a   10Meg
Cgs   g2  s2  0.75n
.ENDS  IPD30N03S4L_14_L0
```

Fig -1: SPICE model of MOSFET (.lib)

Fig -2: Symbol created from SPICE model (.olb)

C. Open OrCAD capture and create circuit diagram specific to required characteristics:

Open design entry tool of PSpice i.e. OrCAD and create circuit diagram using created symbol for required characteristics as shown in Fig-3. Then create simulation profile in which attach .lib file of component which is used in the circuit. Also set DC sweep analysis by giving voltage or current ranges required for that characteristics.

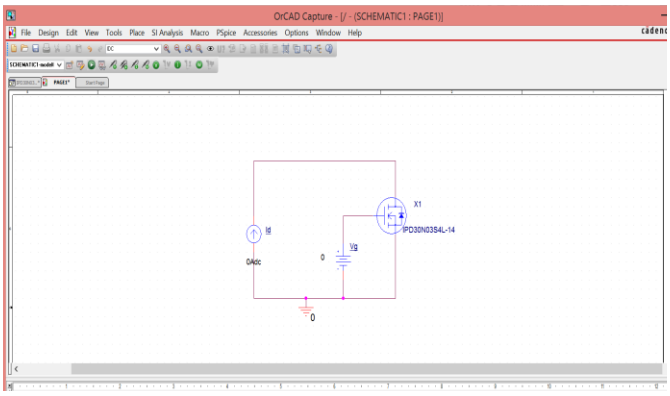

Fig-3: Circuit creation in OrCAD capture

D. Insert voltage or current probes and run the simulation ,then obtain verification result:

By inserting voltage or current probes and running the simulation, obtain the simulation graph for required characteristics. Download the datasheet of that component and compare the datasheet graph values with simulation graph values. Create excel sheet which contain simulation and datasheet graph values, then find the difference between them as,

$$\textbf{Difference}=\frac{(Datasheet)-(Simulation)}{(Datasheet)}*100$$

3. MOSFET VERIFICATION

The Metal Oxide Semiconductor Field Effect Transistor (MOSFET) is a type of Field Effect transistor (FET). Gate voltage determines the conductivity of the device. MOSFETS are very much the device of choice in modern circuit design, be it analogue or digital. Different manufacturers of MOSFET are Infineon Technologies, ON Semiconductor, NXP Semiconductors, and Vishay etc.

MOSFET is characterized by different characteristics such as Power dissipation, Safe operating area, typical output characteristics, typical transfer characteristics, Drain-source on state resistance, Forward diode voltage characteristics, and Avalanche characteristics etc. which are present in datasheet of MOSFET. From all these in SPICE model verification three characteristics are obtained in OrCAD and are compared with datasheet characteristics. These characteristics along with required circuit diagram for verification are explained below:

3.1 Typical Output characteristics (V_{DS}VsI_D):

This characteristic gives plot for Drain current as a function of Drain-Source voltage at given Gate-Source voltage (V_{GS}) value and chip temperature Tj of 25 °C.

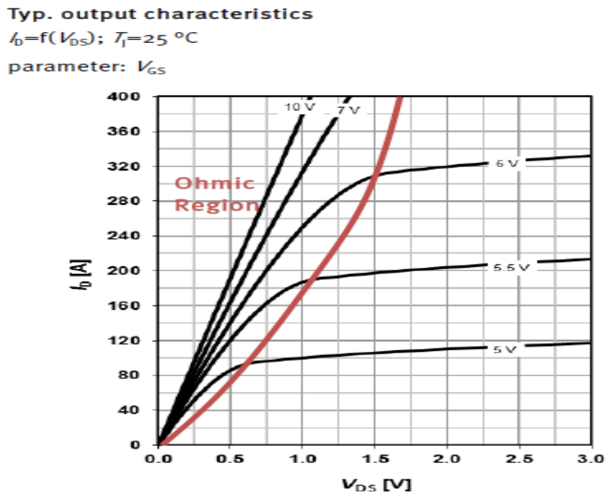

Fig-4: Typical Output characteristics

Datasheet graph for typical output characteristics is shown in Fig-4. For optimal efficiency the MOSFET should be operated in the "ohmic" region. The boundary line between ohmic and saturation region is defined by

$$V_{DS} = V_{GS} - V_{GS(th)}$$

Drain current of MOSFET saturates beyond ohmic region of MOSFET. As the operating point goes into the saturation region, any further increase in drain current leads to a significant rise in drain source voltage and as a result conduction loss increases. For obtaining this characteristic in simulation create circuit diagram with MOSFET as shown in Fig-5. In simulation setting give DC sweep to V_{DS} and V_{GS} then apply current measuring probe at drain terminal of MOSFET. After running the simulation, graph is obtained as shown in Fig-6.

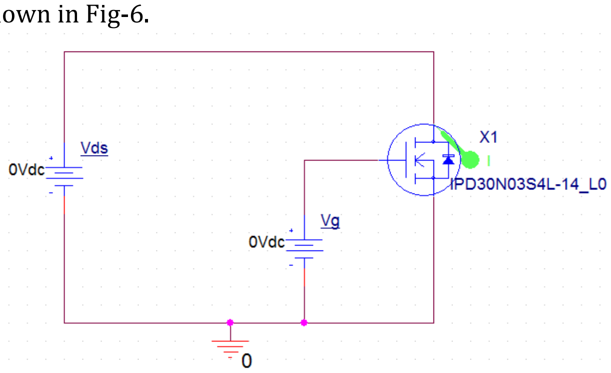

Fig-5: Circuit diagram for typ. Output characteristics

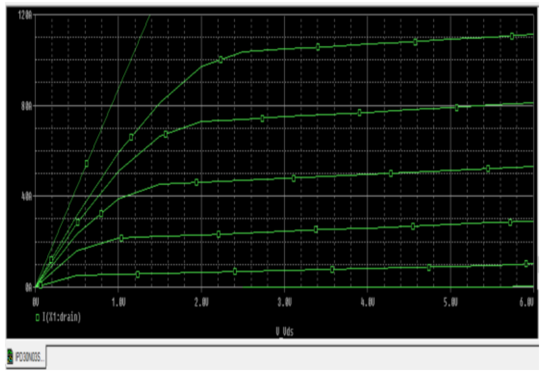

Fig-6: Simulation graph for Typ. Output characteristics

3.2 Typical Transfer characteristics (V_{GS} Vs I_D):

This characteristics gives plot for Drain current as a function of Gate-source voltage at constant V_{DS} at two or three different temperatures as shown in Fig-7.

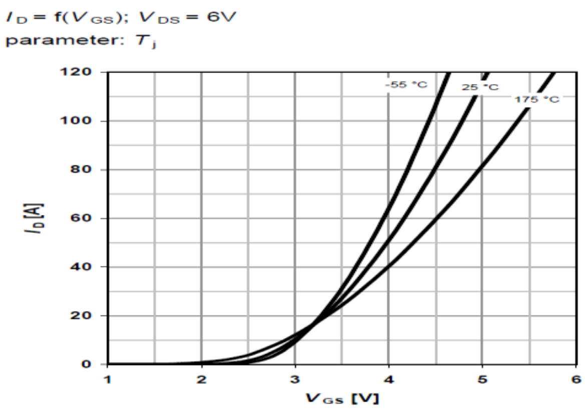

Fig-7: Typical Transfer characteristics

Above graph has curves at -55°C, 25°C and 175°C. All the graphs should intersect at one point, the so called temperature stable operating point. When the gate-source voltage applied to the MOSFET is below this point the MOSFET operates with a positive temperature coefficient, meaning with increasing junction temperature the drain current will also increase. Operating at this condition with constant V_{GS} should be avoided due to possibility of thermal runaway. Due to inability of SPICE model of MOSFET to change behaviour with respect to temperature, simulation graph is only obtained at junction temperature of 25°C. For obtaining this characteristics create circuit diagram as shown in Fig-8. In simulation setting give DC sweep to V_g and apply current measuring probe at drain of MOSFET.

After running simulation graph is obtained as shown in Fig-9.

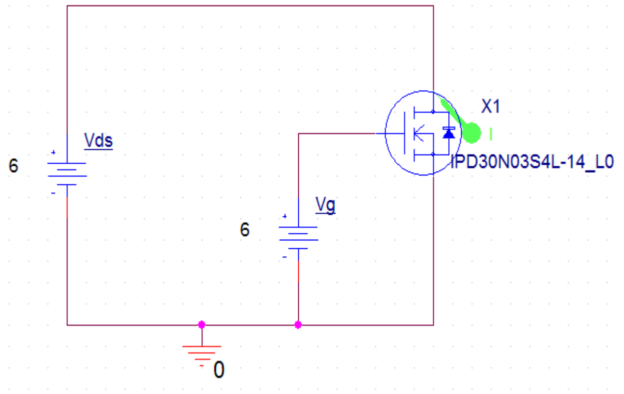

Fig-8: Circuit diagram for typ. Transfer characteristics

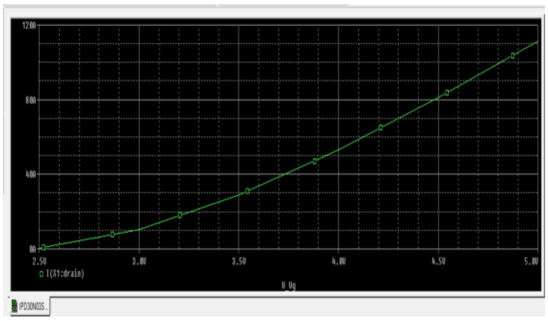

Fig-9: Simulation graph for Typ. Transfer characteristics

3.3 Drain-Source On state resistance Vs Drain current (I_DVs $R_{DS(ON)}$) :

This characteristics gives plot for Drain-Source in state resistance as a function of Drain current at given Gate-Source voltage (V_{GS}) value and temperature T_j of 25 °C. (Fig-10)

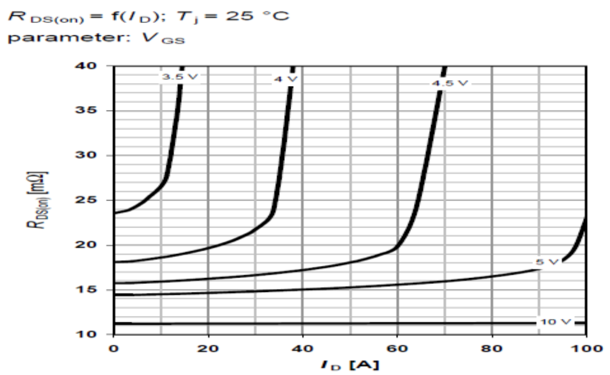

Fig-10: I_DVs $R_{DS(ON)}$ graph

The on-resistance curves change tremendously while a different level of V_{GS} is applied. In below graph to fully turn on a device, V_{GS} of 10V is required. $R_{DS(ON)}$ Value is used to calculate the conduction power loss of the MOSFET. Rdson can be calculated using ohms law as,

$$R_{DS(ON)}(I_D) = \frac{V_{DS}}{I_D}$$

For obtaining this characteristics create circuit diagram as shown in Fig-11. In simulation settings give DC sweep to current source and gate-source voltage. Run the simulation and add trace for obtaining $R_{DS(ON)}$, simulation graph is obtained as shown in Fig-12.

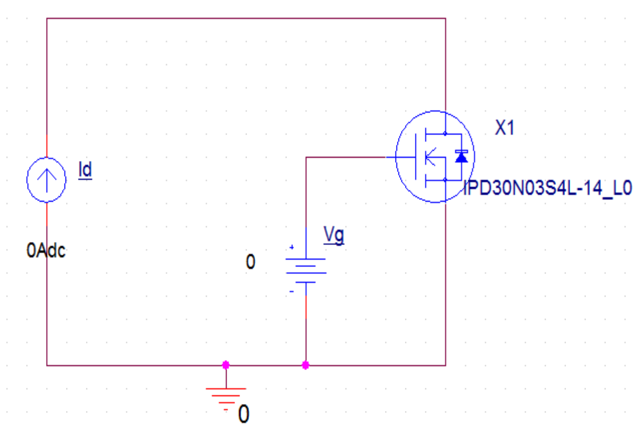

Fig-11: Circuit diagram for I_DVs $R_{DS(ON)}$

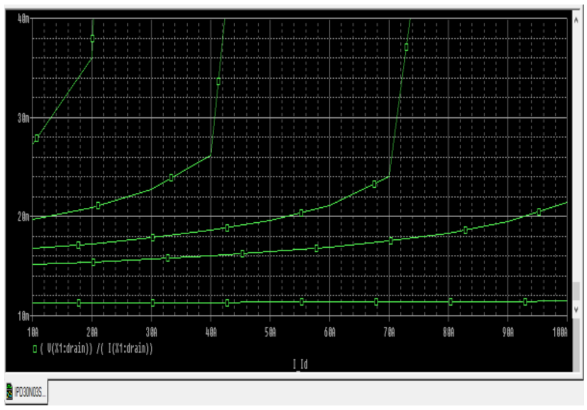

Fig-12: Simulation graph for I_DVs $R_{DS(ON)}$

4. CONCLUSION

We have developed workbench for verification of accuracy of SPICE model of MOSFET. By knowing the accuracy of SPICE model, we can use that accurate model in our PCB level circuit simulation to avoid under and overdesign. Hence verification of SPICE model is necessary for obtaining correct simulation result. This verification also gives behavior of our component at different voltages and currents.

ACKNOWLEDGEMENT

It is a genuine pleasure to express my deep sense of thanks and gratitude to my guide Prof. Amutha Jeyakumar for her exemplary guidance. I am highly indebted to Mr. Devendrabhai Patel, Technical Specialist (Electrical Design), Cummins India Pune, for his guidance and constant supervision as well as for providing necessary information regarding the project & also for his support in completing the course of thesis. I would like to express my gratitude towards my parents & members of Cummins Inc. for their kind co-operation and encouragement which helped me in completion of this thesis.

REFERENCES

[1] Infineon OptiMOS Power MOSFET Datasheet Explanation www.infineon.com.
[2] Gajab, Karna D., and Ajay Kanth Chitturi. "A Functional Analysis of SPICE Simulations and Parameters." International Journal of Science and Engineering Investigations 2.18 (2013).
[3] PSpice A/D Reference Guide Product Version 16.6 October 2012
[4] Sedra and Smith, Microelectronic Circuits, fourth edition, Oxford University Press, 1998
[5] Buttay, Cyril, et al. "Model requirements for simulation of low-voltage MOSFET in automotive applications." IEEE transactions on power electronics 21.3 (2006): 613-624.
[6] Semiconductor Modelling in SPICE Course homepage: http://www.imperial.ac.uk/people/paul.mitcheson/teaching]