Machine Learning-Based Approximation of MOSFET DC and Small-Signal Characteristics from LTspice Simulation

Wenbo Wu

{2960167W@student.gla.ac.uk}

Glasgow College, University of Electronic Science and Technology of China, Chengdu, 611731, China

Abstract. Accurate modeling of MOSFET characteristics is essential for analog circuit design, where designers commonly rely on SPICE simulations to predict transistor behavior under different bias conditions. However, repeated SPICE simulations can become computationally expensive during design exploration and optimization. This study investigates the feasibility of using simple regression-based machine learning models to approximate MOSFET Direct Current (DC) and small-signal characteristics obtained from LTspice simulations. Linear regression and second-order polynomial regression are used to model the relationships between the gate-to-source voltage (VGS) and two outputs: drain current (Id) and small-signal voltage gain (Av). Results show that polynomial regression achieves very high fitting accuracy. For the Id–VGS relationship, the coefficient of determination is approximately one, indicating that the quadratic model closely matches the simulated data within the studied operating range. For the Av–VGS relationship, the coefficient of determination improves from 0.920 to about 0.990, while the RMSE decreases from 1.53 dB to 0.55 dB.

Keywords: Machine Learning; LTspice Simulation; MOSFET.

1. Introduction

With the continuous scaling of semiconductor technology and the increasing complexity of integrated circuits, accurate device modeling has become a fundamental requirement in analog and mixed-signal circuit design [1]. In recent years, machine learning techniques have also attracted growing attention in electronic design automation, where they have been explored for tasks such as modeling, prediction, and design acceleration [2]. Among these approaches, neural-network-based methods are particularly effective for modeling complex nonlinear relationships[3].

In actual circuits, people get our transistor properties from simulations done by SPICE like LTspice, and it gives us good guesses no matter what the biases are [4,5]. As for approximation issues brought

ICIAAI 2026, May 22-24, Singapore, Singapore

DOI 10.4108/eai.22-5-2026.2365246

about by data: it is quite simple and straightforward to use classic regression methods such as polynomial. In the implementation stage people can take use of python's scikit – learn libraries that has been standardized to carry out all regression and quantitation [6].

Regarding device-level modeling, the Berkeley Short-Channel IGFET Model (BSIM) is one of the most widely used compact models for accurate circuit simulation [7]. In the past few years other studies have applied machine learning techniques to MOSFETs as well, proving that transistor models could still be approximated with data-based methods [8]. And Machine learning has also been applied to compact MOSFET device modeling [9], demonstrating the feasibility of data-driven methods at the transistor level [10]. But most of the current work is about more complicated model or bigger dataset so it might not be applicable for a beginner's feasibility study.

Objective of this paper is to find out whether simple regression-based machine learning model can approximate the SPICE MOSFET characteristics on small datasets. Not chasing after super complex architectures, instead making one whole, repeatable flow which contains making & prepping the data, training the model, assessing it and doing some visuals. The two transistor characteristics considered here are the drain-current as a function of V_{GS}, and the small-signal voltage gain as a function of V_{GS}. The aim is to compare the Linear Regression model with a Second Order Polynomial Regression model in order to see if simple machine learning techniques can sufficiently represent the non-linearity of transistor characteristics

2. Related Background and Knowledge

MOSFET is the basic building block of today's integrated circuits and their I-V characteristics are usually modeled by analytical expressions obtained from semiconductor physics [1]. In the saturation region, the current through the drain is about the square of the effective gate overdrive voltage, and the small signal parameters like transconductance. Output Resistance determines the voltage gain of an amplifier circuit[1]. Although the compact model like BSIM gives good results over a large operating range, it is very complex nonlinear equation and it needs to be iteratively numerically solved using SPICE simulator [4].

Or people can think about machine learning as another kind of treatment device like. If people have some set of inputs say bias voltages and geometries, along with the corresponding output, for example, drain currents or gains, then a machine learning (ML) model would learn this mapping from inputs to outputs without knowing any physical equations. The most basic one is linear regression, which assumes that there is a linearity. In polynomial regression people use the idea of adding higher order terms, which can allow for non-linearity [5]. Although neural networks are able to get very flexible as well, simpler regression models should suffice to show the basic feasibility of a model.

This project uses the linear regression and second order polynomial regression because these two are very easy to implement and is good for small dataset. These are baselines to see if people can get close enough to an SPICE result using a ML model before trying other approaches.

3. Data and Method

3.1 Data

Papers data set is coming out of doing an LTspice simulation on a MOSFET Amplifier circuit. V_{GS} goes from 0.80 V all the way up till 1.60 V, every time it steps by 0.05 V. For every operating point people record some properties of our circuits such as the drain current (I_d,), power consumption(P) and small signal voltage gain(A_v).

So people have a total of 17 samples describing how the output current is related to changes in the input voltage V_{GS}.The resulting dataset contains 17 samples that describe the relationship between the input voltage $V_{GS}$and the corresponding circuit responses. These SPICE simulation results are used as the reference values for evaluating the regression models.

The dataset used in this study is summarized in Table 1, which lists the LTspice simulation results for different values of the gate-to-source voltage.

Table 1 LTspice simulation results

VGS (V)	Id (μA)	P (mW)	Av (dB)
0.8	5	0.025	-6.02
0.85	11.25	0.056	-2.5
0.9	20	0.1	0
0.95	31.25	0.156	1.94
1	45	0.225	3.52
1.05	61.25	0.306	4.86
1.1	80	0.4	6.02
1.15	101.25	0.506	7.04
1.2	125	0.625	7.96
1.25	151.25	0.756	8.79
1.3	180	0.9	9.54
1.35	211.25	1.056	10.24
1.4	245	1.225	10.88
1.45	281.25	1.406	11.48
1.5	320	1.6	12.04
1.55	361.25	1.806	12.57
1.6	405	2.025	13.06

3.2 Method

To approximate the relationship between the input voltage $V_{GS}$and the circuit characteristics, two regression models are employed: linear regression and polynomial regression.

Linear regression assumes a linear relationship between the input variable and the output variable. The model can be written as

$$y=\beta_0+\beta_1 x \tag{1}$$

where x represents the input variable V_{GS}, y represents the predicted output (such as I_d or A_v), and β_0 and β_1 are the regression coefficients.

However, transistor characteristics often exhibit nonlinear behavior. To better capture this relationship, polynomial regression is also applied. In this study, a second-order polynomial model is used:

$$y=\beta_0+\beta_1 x+\beta_2 x^2 \tag{2}$$

This model introduces a quadratic term that allows the regression function to approximate nonlinear relationships between the input and output variables. The performance of the regression models is later evaluated by comparing the predicted values with the SPICE simulation results.

4. Experimental Results and Discussion

4.1 Modeling of Id–VGS Characteristics

The Id–VGS characteristic exhibits a strong nonlinear relationship due to the square-law behavior of the MOSFET in saturation [1]. Figure 1 compares the fitted curves obtained using linear regression and second-order polynomial regression with the original LTspice data.

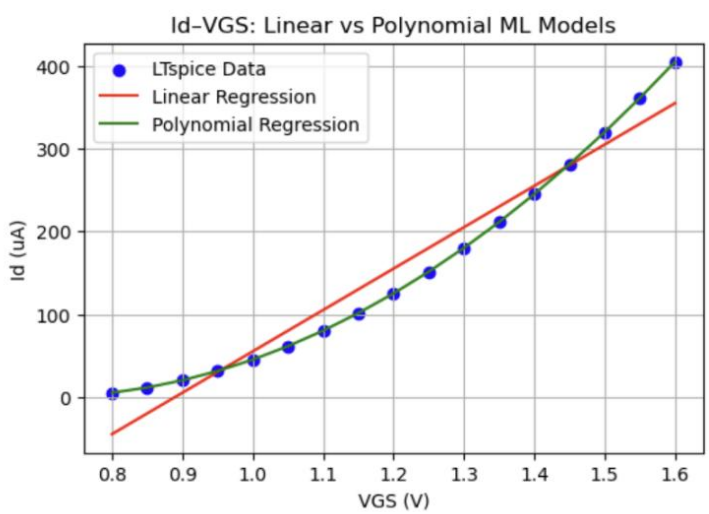

Figure 1. Comparison of linear and polynomial regression for Id–VGS modeling. Source: Original.

It's also seen in the linear regression does not capture the curve of the Id-VGS when it gets higher gate voltage. On the contrary, the polynomial is sticking with all along its original data over this whole voltage range. The quantitative judgement is right. The linear model gets an R^2 of 0.9547 and the polynomial model achieves a perfect match for the data since it has an R^2 of 1 along with a zero MAE & RMSE.

To show the prediction accuracy, the figure is scatter plot between real and predicted Id value in Figure 2.

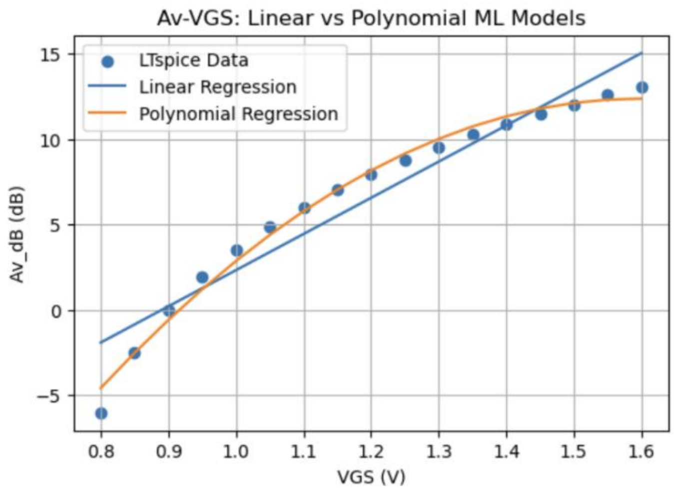

Figure 2. True versus predicted Id values for linear and polynomial regression models. Source: Original.

Polynomial model points all sit right on the ideal diagonal, this verifies that it fits really well. It can be seen from this, that a simple second order model is enough to properly approximate the Id-VGS characteristic for fixed bias.

4.2 Modeling of Av–VGS Characteristics

The small-signal voltage gain Av exhibits a nonlinear dependence on VGS due to variations in transconductance and output resistance [1]. Figure 3 Fitted curve of Av-VGS by using Linear regression and Polynomial regression.

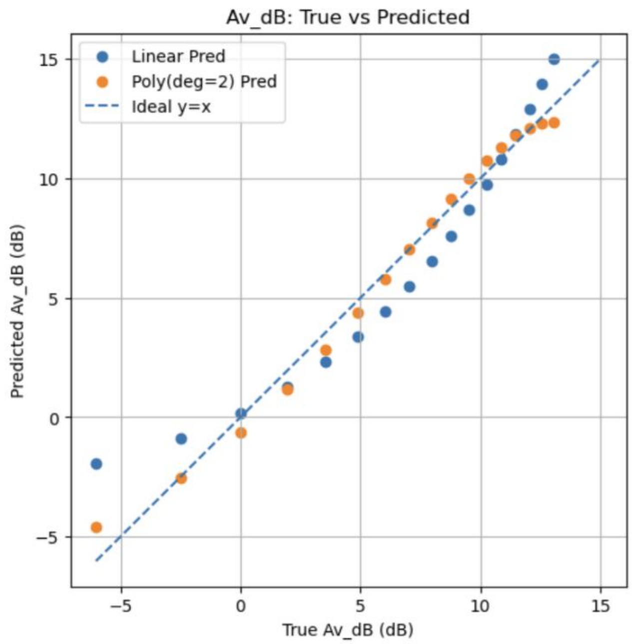

Figure 3. Comparison of linear and polynomial regression for Av–VGS modeling. Source: Original.

Av-VGS is even harder to model as compared to the Id - V GS case. Linear regression model can only make a rough estimate and polynomial regression model is more accurate than previous one. Performance metrics show that the linear model achieves an R^2 value of 0.9204, whereas the polynomial model improves the R^2 value to 0.9896 and reduces the RMSE from 1.5272 dB to 0.5531 dB.

Figure 4 gives the true and predicted Av values for both the models.

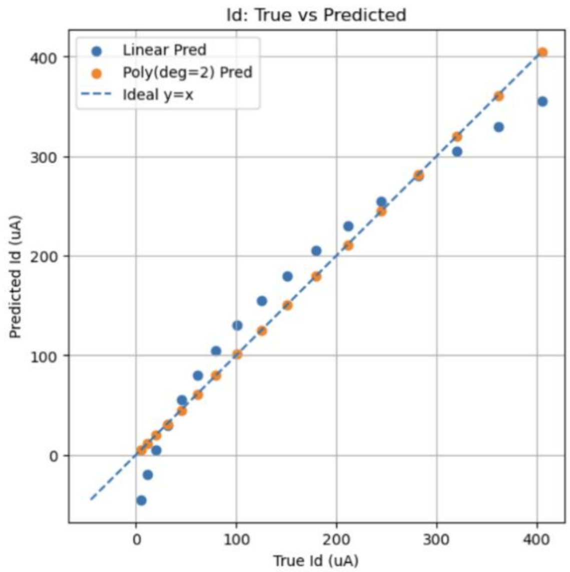

Figure 4. True versus predicted Av values for linear and polynomial regression models. Source: Original.

Polynomial model predictions are packed near the ideal line, but the linear model is off from the ideal line in low and high gain area. The above result shows that using polynomial regression to model the nonlinearity of small-signal gain is better than.

4.3 Discussion

From experiment if people use a Simple Regression Based ML model the transistor characteristics generated by SPICE could be predicted if all other operating condition(s) are kept constant and Dataset is well-behaved[2,3][6]. Polynomial regression is much better than linear regression for both Id-VGS and Av–VGS: This demonstrates that it's important to have nonlinearities in the model structure. The Id-VGS fits were performed perfectly, this suggests the true nature of the relationship can be described as a quadratic function over the given range.

And also note that in our feasibility study here, all available data points are used for training, and no separate test set is introduced. It's fine to check for model and workflow basics here, but doesn't say anything about being able to generalize. In reality we'd need larger datasets and proper training–validation splits. However, it is clear from the present result that ML is a good candidate to model transistor behavior.

5. Conclusion

The study investigated if it's possible to learn the SPICE simulated MOSFET characteristic by using simple linear or polynomial regression model. A small dataset is obtained from LTspice by sweeping the gate-to-source voltage V_{GS} and recording the corresponding drain current I_d and small-signal voltage gain A_v. Apply two regression approach i.e., linear regression & second order polynomial regression in order to find out the relation between input voltages with the circuit response. The purpose is to find out if lightweight machine learning model can learn about the transistor very well.

It is shown that the polynomial regression gives an excellent approx. of the SPICE sim. outputs: For the case of I_d-V_{GS}, polynomial model has almost an R^2 of 1 and very low prediction error. Regarding the A_v-V_{GS} relationship, it is obvious that the polynomial model predicts better than a linear model. Besides. This study set up whole workflow links LTspice simulate, orderly arrange collected dataset, do some regressions, then make a little assessment. People see that even without lots of data, a machine learning model can be pretty close to the transistors.

The future work could go even deeper in more complex ones. Like adding more inputs like V_{DS}, and the transistors shapes to make more than one parameter model. Furthermore, even with more advanced ML techniques or bigger datasets could be used to generalize even further and perform holistic prediction at the circuit level.

References

[1] Gray, P. R., Hurst, P. J., Lewis, S. H., Meyer, R. G.: *Analysis and Design of Analog Integrated Circuits*. 5th edn. Wiley (2009).

[2] Rapp, M., Amrouch, H., Lin, Y., Yu, B., Pan, D. Z., Wolf, M., Henkel, J.: MLCAD: A Survey of Research in Machine Learning for CAD Keynote Paper. IEEE Transactions on Computer-Aided Design of Integrated Circuits and Systems. 41(10), pp. 3162–3181 (2022). DOI: 10.1109/TCAD.2021.3124762.

[3] Haykin, S.: Neural networks and learning machines. 3rd edn. Pearson (2009)

[4] Analog Devices: LTspice Simulator. https://www.analog.com/en/resources/design-tools-and-calculators/ltspice-simulator.html, accessed 30 July 2026.

[5] Hastie, T., Tibshirani, R., Friedman, J.: The elements of statistical learning. 2nd edn. Springer (2009)

[6] Pedregosa, F., et al.: Scikit-learn: Machine learning in Python. *Journal of Machine Learning Research*. 12, pp. 2825–2830 (2011).

[7] Hu, C., Liu, W.: *BSIM4 and MOSFET Modeling for IC Simulation*. World Scientific (2011).

[8] Tung, C.-T., Kao, M.-Y., Hu, C.: Neural network-based I–V and C–V modeling with high accuracy and potential model speed. *IEEE Transactions on Electron Devices*. 69(11), pp. 6476–6479 (2022). DOI: 10.1109/TED.2022.3208514.

[9] Wei, J., Wang, H., Zhao, T., Jiang, Y.-L., Wan, J.: A new compact MOSFET model based on artificial neural network with unique data preprocessing and sampling techniques. *IEEE Transactions on Computer-Aided Design of Integrated Circuits and Systems*. 42(4), pp. 1250–1254 (2023). DOI: 10.1109/TCAD.2022.3193330.

[10] Yang, Y.-S., Li, Y., Kola, S. R. R.: A physical-based artificial neural networks compact modeling framework for emerging FETs. *IEEE Transactions on Electron Devices*. 71(1), pp. 223–230 (2024). DOI: 10.1109/TED.2023.3269410.