Retraction

Retraction: DC Motor Simulation using LTSpice (IOP Conf. Ser.: Earth Environ. Sci. 426 012137)

Published 08 December 2021

This article has been retracted by IOP Publishing following an allegation that this article reproduces FesZ Electronics (2019, March 29) LTSpice Tutorial - Modeling a DC brushed motor [Video]. YouTube: https://youtu.be/Wc4XzTrWSpo. IOP Publishing has investigated in line with the COPE guidelines, and the author agrees they reproduced this work without credit or acknowledgement. Consequently, this paper has been retracted by IOP Publishing. The authors agree to this retraction and apologise to the original creator of the work.

Retraction published: 08 December 2021
DC Motor Simulation using LTSpice

J Linggarjati
Computer Engineering Department, Faculty of Engineering, Bina Nusantara University, Jakarta, Indonesia 11480

Corresponding Author: jimmyl@binus.edu

Abstract. DC motor is simulated in this paper, in which LTSpice simulator is used. DC Motor is still highly needed in industrial world, for applications that needs goods’ movement in the process line. Therefore, in this paper, it is important to do simulation before real-world electronic circuit is developed. Thereby we promote green eco-friendly environment, by reducing electronic waste caused by prototyping. The simulation of DC Motor is divided into three (3) parts. First, a simplified DC motor model is assumed, and then the effect of inductance and inertia of the DC motor are considered. Lastly, a small perturbation is noticeable in the output of the electrical DC motor, and after a careful insight into the slip-ring mechanical inside of the DC motor, the effect of it is added in the final simulation model. This gradual modelling methodology certainly helps students to correlate between DC motor modelling with the sampled real-world data, which is taken directly from the experimental laboratory.

Keywords: DC Motor, LTSpice, Simulation.

1. Introduction
In industrial world, the controlled DC Motor is needed for various automation applications, such as one in plastic flying shear machine, as was written in [2]. Therefore, in this paper, an LTSpice electronic circuit simulation [1] for understanding how a DC motor operates, is developed.

LTSpice simulator is developed by analog devices [3]. And as most of the electronic simulator, it is a derivative product of SPICE [4]. PSpice has also been used for simulation of electro-mechanical devices, such as induction motor in industrial application, as written in [5]. LTSpice has been used widely in electrical and electronic students around the world, such as the following report written in [6].

Permanent Magnet DC Motor (PMDC) is a class of motor in which the stator has a fixed field, due to the permanent magnet attached to it. This type of motor has a linear characteristic with regards to its excitation voltage at the rotor. And therefore, the control of speed can be done, by adjusting the voltage terminal at the rotor field.

2. Simulation Model of DC Motor
Based on Figure 1, the following formulas are used:

\[ v = Ri + L \frac{di}{dt} + V_e \]  

(1)
\[ T = J \frac{dv}{dt} + T_{\text{load}} + bw \]  

(2)

Figure 1. Permanent Magnet DC Motor Circuit

2.1. Simplified model of DC Motor in LTSpice

In order to understand the basic principle of DC Motor, the model of DC Motor is simplified to contain only the resistance of its coil, as shown in Figure 2.

The circuit diagram in Figure 2, models the DC Motor as it is only consisting of pure resistance only. And to model the magnetic coupling between the electrical part of the motor and the mechanical part of the motor, we used dependent voltage source of type CCVS (Current Controlled Voltage Source), as shown in Figure 2, as B1, B2, B3, and B4.

B1 models the back-emf (electromagnetic force), which is a voltage that emerges as the rotor spins. Therefore, the bemf voltage is controlled by the speed of the rotor. And in this simulation, it is modeled as current which exist in B2. While B2 is a voltage source that model the mechanical torque being developed as a result of magnetic coupling between rotor and stator. The current of B2 is the value that determines the speed of the rotor, and it is controlling B1 in the form of bemf voltage. Therefore, the equation 1, is modeling between electrically bemf and the mechanical speed that caused B1.

\[ V = -I(B2) \]  

(3)

Notice that the minus sign is because the polarity of \( V_{\text{bemf}} \) always opposed with the motor voltage source. If we look from the mechanical side, B2 models the torque produced by the magnetic coupled. The torque value depends also with the current at B1, so it can be concluded that torque is decreased...
as the speed of DC motor increased. This happens because the $V_{\text{bemf}}$ is approaching $V_{\text{source}}$, causing the current to decrease, as shown in Equation 4.

\[ \frac{v - V_{\text{bemf}}}{R + Ls} = I(s) \]  

Equation 4 is corresponding nicely with the graphic characteristic of PMDC motor, as shown in Figure 3. The vertical axis is speed, while the horizontal axis represents torque. As we give load to the mechanical rotor, a power maximum transfer occurs at maximum torque ($T_s$) divided by 2. The efficiency graph below, further describes the efficiency of the motor input power over the motor output power. And as plotted in Figure 3, we must use the motor operation point below $T_s/2$, otherwise energy heating inside the motor is wasted.

2.2. *Inductance and Inertia in DC Motor Modelling*

The second step toward realistic DC Motor modelling in LTSpice software, is to put inductance and inertia as part of the DC Motor Modelling. As we have learned from the physics of inductor, it is considered as a memory component, which means that ideally it consumes no power, and the rate of current change will produce voltage across the inductor, as shown in equation 5. In the mechanical part, we have the motor inertia ($J_m$), being modelled as inductor in LTSpice. And the torque produces by the motor inertia is the inertia multiplied by the rate of angular speed change, as shown in equation 6.

\[ V_I = L \frac{di}{dt} \]  
\[ T_{\text{inertia}} = J \frac{d\omega}{dt} \]  

In Figure 4, we added the inductance to the DC Motor model.
2.3. Commutator Effect in DC Motor Modelling
If we look closely into the design of commutator, it actually switches on and off the voltage source, as the rotor coils rotates through the commutator (as illustrated in Figure 5). Figure 6 shows the circuit that does the modelling of the commutator.

![Figure 4. DC Motor Modelling with R and L](image)

![Figure 5. Commutator Illustration in DC Motor](image)

![Figure 6. Commutator Switch Model in DC Motor Modelling](image)
3. Simulation Result
Here, we discuss the results of modeling DC Motor in LTSpice. The results are divided into 3 gradual steps, to foster better understanding of how a DC Motor works.

3.1. Simplified model of DC Motor in LTSpice
By using the circuit modelling of Figure 2, we can see that the simulation output for speed and back-emf voltage are exactly following the voltage source, as shown in Figure 7. Notice that the back emf voltage is approximately 2 volts less than the voltage input (12 volt). This is due to the mechanical load applied, which is simulated in B4 (in Figure 6).

![Figure 7. Motor Speed (a) and V_{bemf} (b)](image)

3.2. Inductance and Inertia in DC Motor Modelling
Referring to circuit simulation in Figure 4, a more realistic DC Motor behaviour is achieved. This is shown in Figure 8. The motor speed does not immediately jump to a no-load speed but has a sloped due to the inertia of the rotor. The deceleration speed is also sloped. And if we look at the motor current, it is not flat, but has a spike due to the inductance of the DC Motor.

![Figure 8. Motor Speed (a), Motor Current (b), Back-emf voltage (c)](image)
3.3. **Inductance and Inertia in DC Motor Modelling**

The DC Motor voltage is not flat, as shown in Figure 9. It has spike with a constant interval, dictated by the mechanical slotted ring. And by using simulation circuit in Figure 6, we can simulate the disrupted effect from the commutator ring.

![Figure 9. Commutator Effect on V\text{motor} (a) and I\text{motor} (b)](a)(b)

4. **Conclusion**

LTSpice as a free and yet powerful electronic simulator, has great beneficial for students to do simulation before they build the circuit prototype. This has promoted high productivity for students and promote green environment by reducing garbage prototyping.

The simulation result of a permanent magnet DC motor has been shown in Figure 8 and 9. And it has been confirmed with the real experimental data shown in the oscilloscope. Therefore, this simulation result is a better PMDC Modelling motor, which can be further used in control system where students will encounter for their next study in control engineering courses.

The LTSpice has also helped students in understanding theory of DC Motor by comparing the simulation result with the experimental data. Future work will further investigate real DC Motor characteristic and then used their data us input to the simulation circuit.

**References**

[1] Jenni F. 2006 Simulation tool (CERN-2006-010) Paul Scherrer Institute, Villigen, Switzerland. *CAS - CERN Accelerator School and CLRC Daresbury Laboratory: Specialised CAS Course on Power Converters*, Warrington, UK, 12 - 18 May 2004. 363-384.

[2] Wu H., Wang C. and Zhang C. 2014 Design of Servo Controller for Flying Shear *Journal of Networks*, **vol. 9, no.** 11, p. 3038.

[3] LTspice [Online]. Available: https://www.analog.com/en/design-center/design-tools-and-calculators/ltspace-simulator.html.

[4] Pederson D. 1984 A historical review of circuit simulation *IEEE Transactions on Circuits and Systems* **vol. 31, no.** 1, pp. 103-111.

[5] Schiop A. and Popescu V. 2007 Pspice simulation of power electronics circuit and induction motor drives *Rev. Roum. Sci. Techn. -Electrotechn. et Energy* **vol. 52, no.** 1, pp. 33-42.

[6] Lutovac M. and Mladenovic V. 2015 Contemporary electronics with LTSpice and mathematica *International Scientific Conference of IT and Business-Related Research*, Belgrade, Serbia, pp. 134--138.

[7] Da Silva G. F., Marques S. T. and Faria A. W. C. A Reliable Computational System For Estimate The Performance Curve Of PMDC Motors.