



Numerical Analysis of Linear Elevator Structure Using Finite Element Method

Saba A. Othman^{*}, Jamal A.-K. Mohammed, Farag M. Mohammed

Electromechanical Engineering, University of Technology, Baghdad, Iraq.

^{*}Corresponding author Email: eme.19.27@grad.uotechnology.edu.iq

HIGHLIGHTS

- The maximum von- Mises stress and the deformation of elevator frame be acceptable.
- The maximum deformation of car base plate be acceptable, which less than $h/5$ that satisfy the design boundary condition of $\omega < h/5$.
- The maximum stress and deformation that occur on the plate and frame of elevator present save design as compared with the mechanical properties of the materials' selection.

ABSTRACT

In this work, the structure of the linear elevator prototype had been investigated numerically using finite element method. The linear motor structure parameters analyzed using Maxwell ANSYS. The time-stepping method depending on Maxwell equations be applied for analyzing and optimizing the magnetic and force characteristics. While the elevator structure parameters were analyzed using ANSYS workbench based on the principle of virtual work. The frame considered as clamped- clamped beam, and the base of the car considered as thin plate with small deflection. The analysis done with maximum applied load of 360 N at 1.5 safety factor. The results show the distribution of the magnetic lines, the flux density values plus the leakage flux inside the slots. The maximum Von-Mises stress and the deformations of the frame and plate at maximum load are acceptable and present save design. In which the maximum deflection of the thin plate not exceed (thickness/5) at maximum design load.

ARTICLE INFO

Handling editor: Muhsin J. Jweeg

Keywords:

Elevator Structure
Linear Synchronous Motor
Stress
Deformation
Finite Element Method

1. Introduction

An elevator is a mechatronic system used to move passengers and goods safely, swiftly, and comfortably in buildings [1]. The elevator is also essentially a platform that is pushed up by a mechanical means and consists of a car mounted on a platform within an enclosed space called a Hostway. It is very difficult to predict the passengers' location entered into the elevator car and their distributions in regular operation. Loads imposed on the members of the car sling are more complicated than other elevator equipment because there is a different kind of loads that go to the car frame. The elevator car frame is an essential component as safety conditions must be taken into account. On this basis, several researchers focused on this aspect:

Janovsky [2] discussed the stress-determining methods in individual frame parts of the car. Solmazog Lu and Akisin [3] showed a comparison between the traditional calculation results of the freight elevator and the collected findings when the elevator is exposed to an individual-side operation of the safety gear. Onur and Imrak [4] had suggested the construction of frame-shaped steel for stress analysis and performed calculations for displacement and stress, taking into account the distribution of load on the car platform. Bablick et al. [5] used the finite element method (FEM) and the traditional method for investigating the suspension of the car frame system behavior. A dynamic model was developed by Feng et al. [6] according to the theory of solid body dynamics of the elevator car in 3-dimensions. They introduced the modeling and the power control for a high-speed elevator under horizontal vibrations. The FEM is one of the most accepted and widely used tools for the solution and optimization of linear and nonlinear partial differential equations. Mohammed et al. [7] applied MAXWELL ANSYS

software based on the electromagnetic field FEA method to optimally design a Permanent magnet linear synchronous motor (PMLSM) and analyze its dynamic characteristics and parameters.

Raghda'a, Farag and Jamal [8], A Maxwell ANSYS simulation was performed to study and enhance the dynamic characteristics and performance of the model. The results of the improved model design showed that the maximum value of the force ripple is reduced by about 81.13% as compared with the primary model at a smaller ripple coefficient of 0.22%. Saba, Jamal and Farag [9], investigate the Linear Elevator with High Riding Quality Based on S-Curve Profile. The results show that using the driver leads to reduce the jerk by about 88% compared to that without using the driver. The motor designed by researchers [7] has been combined in a linear elevator for the present work. A numerical analysis using FEM was done using Maxwell ANSYS to analyze the motor PMLSM structure parameters. While the elevator structure parameters (car and frame) were analyzed using ANSYS workbench based on the principle of virtual work.

The main objective of this study is to investigate the structural behavior of the linear elevator driven by PMLSM using FEM with the help of ANSYS software.

2. Linear Synchronous Motor

The three-phase single-sided flat-type of permanent magnet linear motors PMLSM designed by [7] with slotted core and surface shown in Figure 1 is employed to drive the cabin in the elevator model. The motor core is consisting of 13 openings, 13 teeth, and 12 openings filled with a single layer of three-phase coil winding concentrated into a laminated 1010 steel solid core. Each motor phase has four slots. These magnetic cores were made from thin films to reduce eddy currents flowing through them. The proposed motor has a three-phase and ten-pole which means that the number of translator slots per electrode per phase is 0.4. The wrapping pattern is the two-layer winding of concentrated type and the winding stretch is a partial notch.

The analysis of motor structure parameters was performed using the commercial finite element package based on Maxwell ANSYS depending on Maxwell equations that ignored end windings due to using 2D model [8]. The solution was divided into two stages, load and no-load transient magnetic solution, to analyze and optimize the magnetic and force characteristics of the linear motors LM for solving the nonlinear transient magnetic problem. A Time-stepping method depending on Maxwell equations was applied. The calculated values of motor input voltage are 45V (at load) and frequency of 17.36Hz, whereas the solution would be divided into 36 steps with 1.6 ms time period at 10^{-4} nonlinear residual using the integration method with backward Euler time. Appropriate meshing is essential to get an accurate solution of the model [10]. The present model has 1200 PLANE121 mesh-elements which is a 2-D, 6-node, charge-based electric element as shown in Figure 2. The easiest method to obtain a high air gap mesh density, relatively suitable mesh of the model is by choosing the free triangular option in the mesh settings. This, however, requires some adjustments to get a converging solution. Some areas require a more accurate mesh than others [11,12]. The mesh generation of the original motor is shown in Figure 3. Three regions of meshing, each contains 1200 triangular element shape with 5.76mm maximum element length.

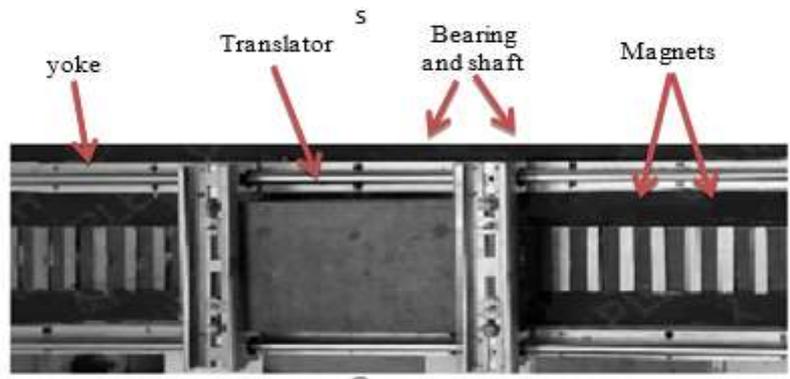


Figure 1: The PM linear synchronous motor prototype [7]

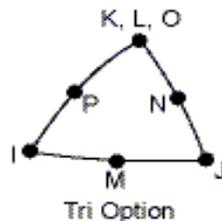


Figure 2: PLANE121 element type

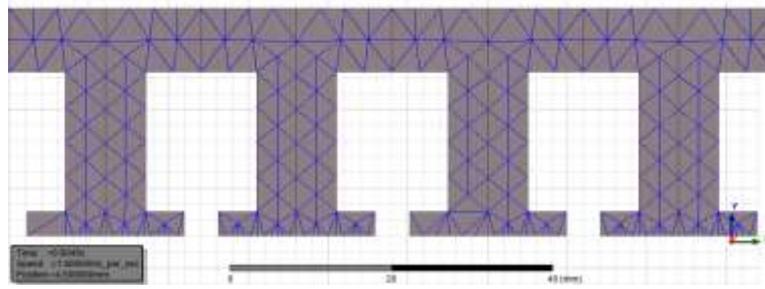


Figure 3: Mesh of one region

3. Analysis of Elevator Structure

The primary concept of the FEM is to estimate the cabin elevator structure, connect elements through finite nodes. According to the virtual work principle, they can be solved by deformation coordination conditions [13, 14].

$$\text{Virtual work of external Force} = T^e = \{\delta\}^e \{F\}^e \quad (1)$$

$$\text{virtual work of stress} = \int \{\epsilon\}^T \{\sigma\} d\Omega \quad (2)$$

$$\text{According to the default work principle, } \{\delta\}^e \{F\}^e = \int \{\epsilon\}^T \{\sigma\} d\Omega \quad (3)$$

By typing the expression of stress and strain in virtual displacement, the stiffness matrix of the element may be obtained. Then the equilibrium equation expressed as follows:

$$\{F\} = [K] \{u\} \quad (4)$$

Each node displacement in the system, the strain and the stress of the corresponding node may be given.

The main elevator structure parameters are consisting of car and frame as shown in Figure (4). The main part of the frame is the four beams, so the load is distributed on them and the analysis will be considered on one. The maximum load on each column or beam takes when the elevator car reaches the highest point, in which the structure subject to the maximum bending moment and buckling. In this work, the proposed elevator frame dimensions are shown in Figure 4. The frame material selected is an iron of 7874 Kg/m³ density, 0.29 Poisson's Ratio, and 204 GPa Young's Modulus.

Since the elevator cabin frame is considered an essential element of elevator safety. But it is very difficult to estimate the location of the travelers who entered the elevator cabin in addition to their distribution in normal operating conditions. Loads imposed on elevator cabin sling members are more complicated than other elevator equipment because different types of load are embedded within the car frame. The base of the car be considered as a flat plate with small deflection in which $b/h > 5$ and $\omega < h/5$, where b and h are the plate width and thickness respectively while ω is the plate deflection. The elevator car material is PVC have Poisson's Ratio 0.38 and Young's Modulus of 2.758 MPa.

In meshing steps, the geometry is divided into small size volume element. Meshing has accomplished with (SOLID186) elements types which they are higher order 3-D solid element having three degrees of freedom per node: translations in the nodal x, y, and z directions shown in Figure 5. Using three dimensions ANSYS software package which consisted of four main steps [15] that:

1. Creating the physical environment:
 - Specify elements type and option.
 - Define the elements coordinate system.
 - Define elements real constant.
 - Specify material properties.
2. Building and mesh the model and assigning physics attributes to each region within the model.
3. Applying boundary conditions and loads.
4. Run solution

There are 938 elements with 6636 nodes for the frame and 660 elements and 609 nodes for the plate, as shown in Figure 6, it solved with finite element method in the solver step, to achieve better accuracy in the solution

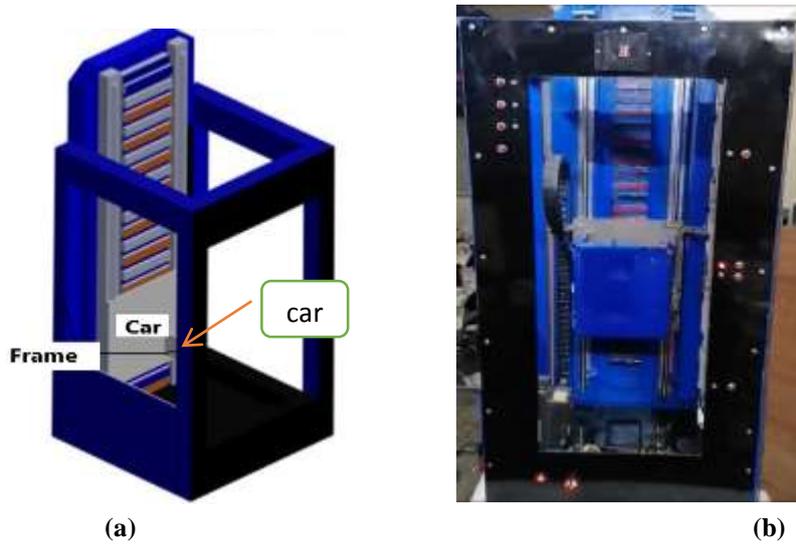


Figure 4: The overall structure of the linear elevator

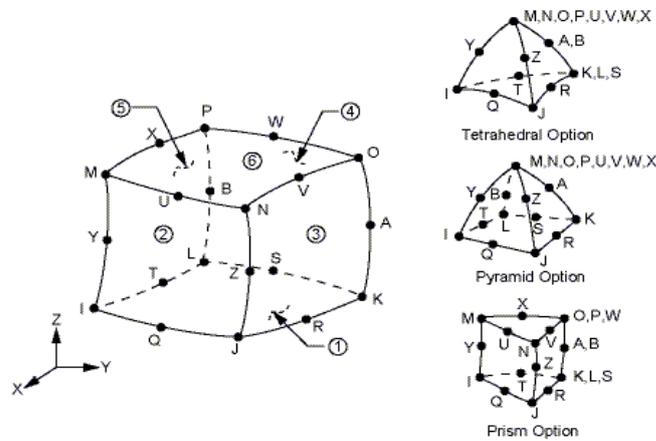


Figure 5: SOLID186 Elements types

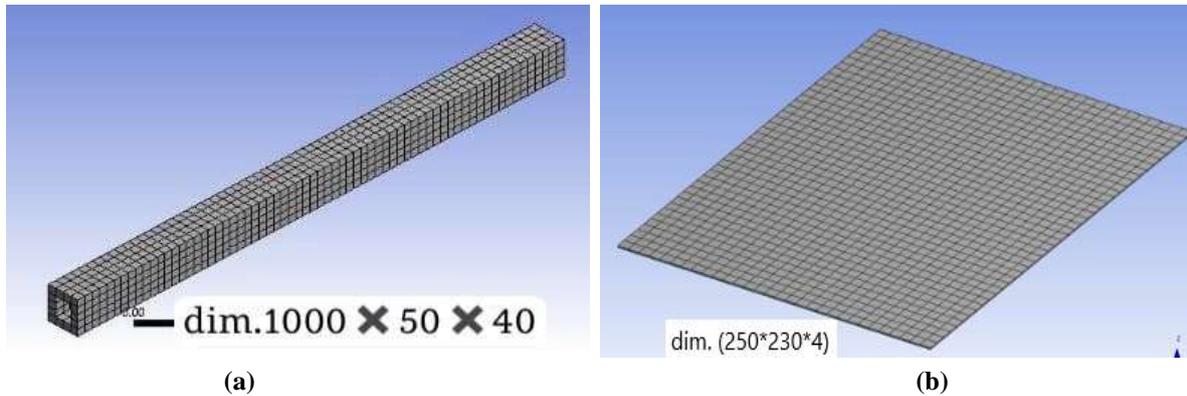


Figure 6: Finite element for (a) Frame (b) Plate

4. Implementations and Discussion

The analysis of motor structure parameters was performed using the commercial finite element package based on Maxwell ANSYS depending on Maxwell equations. The results on the no-load case, considering the axial symmetry of the PMLSM, the designed motor was simplified to a 2D axial symmetry model in Cartesian coordinates ignoring core losses. Irregular distribution of the current density in the conductor was considered. It is known for magnetic components, that the thrust force is heavily affected by magnetic flux density. Finite Element Analysis FEA is a numerical effective tool may be used in this paper to estimate the electromagnetic field density distribution based on geometry and material of machine using Maxwell ANSYS 15.0.0. Figure 7 shows the flux density distribution for the optimum design at the translator iron core, stationary back iron, and the permanent magnets at no load case. The concentration of the flux density in the teeth of the core and the leakage flux inside the slots are clearly noticed. It can be noticed that the magnetic flux is moving in transverse direction with uniformly distribution, which create a better output thrust force with minimum undesired ripple force.

In the frame structures analysis considering the frame as a clamped- clamped structure subjected to different loads depending upon the weight of the persons in the cabin and the location of the cabin. Supported fixed parts have been analyzed using 3D finite element analysis with the aid of ANSYS Workbench. There are many trails to reach the optimal design of proposed elevator frame and car. The elevator operates with maximum load of 240 N at 1.5 safety factor. So, in numerical analysis 360 N applied load be considered in which act at the center of the base plate.

Figure 8 presents the von-Mises stress distribution on the frame subjected to the maximum load when the car reaches the maximum height. The maximum Von-Mises stress value is 2.3006MPa at the lower point of the frame. The maximum deformation of frame occurs at the highest point with 0.075522 mm as shown in Figure 9. The maximum von- Mises stress and deformation of frame indicate the frame design be save at maximum load applied.

Figure 10 shows the distribution of the von-Mises stress on the base plate of the elevator car, the maximum Von-Mises stress is 2.93 MPa, while the maximum deformation of the plate is 0.4934 mm in which occurs in mid of the plate as shown in Figure 11. As the plate thickness is 4 mm, so the maximum deformation of plate be acceptable in which less than $h/5$ that satisfy the design boundary condition of $< h/5$.

The maximum stress and deformation that occur on the plate and frame as compared with the mechanical properties of the materials' selection show a safe elevator structure design with good safety factor.

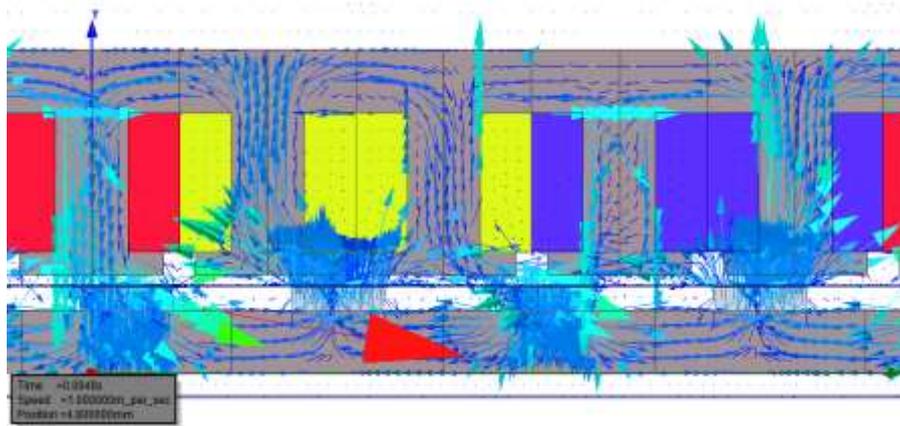


Figure 7: Magnetic field density

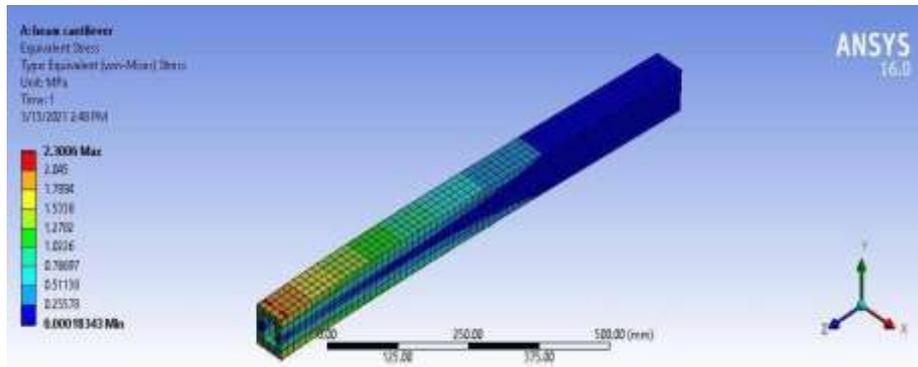


Figure 8: Von-Mises stress distribution.

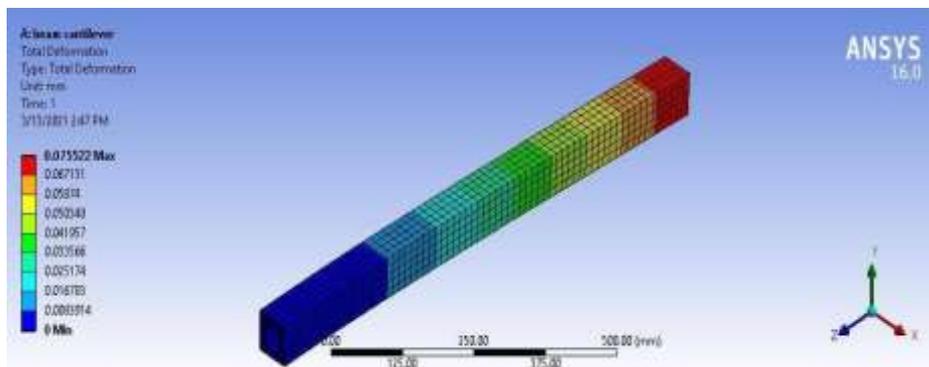


Figure 9: Deformation distribution.

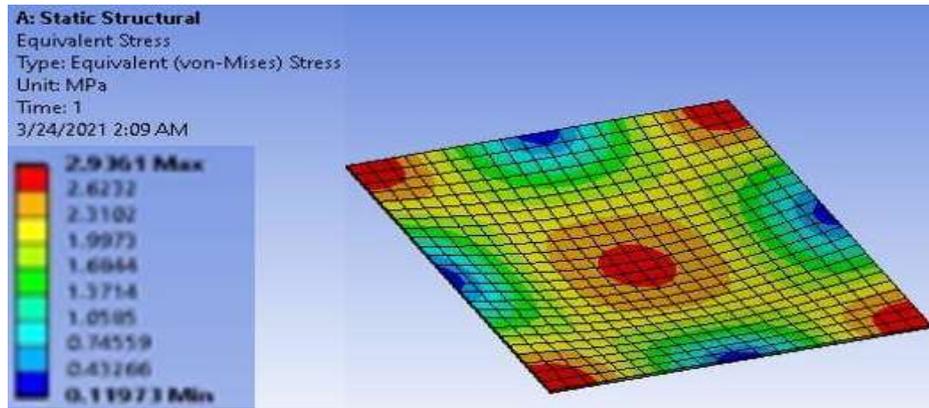


Figure 10: Von-Mises stress distribution on plate

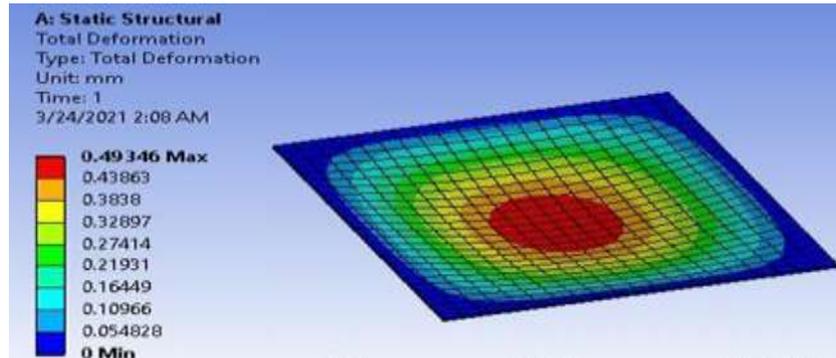


Figure 11: Deformation of plate

5. Conclusions

The finite element method by Maxwell ANSYS was used to analyze the motor structure parameters and ANSYS workbench based on the principle of virtual work to analyzed the elevator structure parameters. The frame be considered as a clamped-clamped beam while the car plate be considered as a thin flat plate with small deflection. The analysis done with maximum applied load of 360 N. The following conclusions can be presented:

- 1) The numerical analysis using Maxwell ANSYS shows the flux density distribution for the optimum design at the translator iron core, stationary back iron, and the permanent magnets at no load case. The magnetic flux is moving in transverse direction with uniformly distribution, which create a better output thrust force with minimum undesired ripple force.
- 2) The maximum von- Mises stress and the deformation of elevator frame be acceptable.
- 3) The maximum deformation of car base plate be acceptable, which less than $h/5$ that satisfy the design boundary condition of $\omega < h/5$.
- 4) The maximum stress and deformation that occur on the plate and frame of elevator present save design as compared with the mechanical properties of the materials' selection.

Author contribution

All authors contributed equally to this work.

Funding

This research received no specific grant from any funding agency in the public, commercial, or not-for-profit sectors.

Data availability statement

The data that support the findings of this study are available on request from the corresponding author.

Conflicts of interest

The authors declare that there is no conflict of interest.

References

- [1] T. Tyni, and J. Ylinen, "Evolutionary bi-objective optimization in the elevator car routing problem," European Journal of Operational Research, 169(3), 2006.
- [2] L. Janovsky, "Elevator mechanical design," New York: Ellis Horwood Series, 1993.
- [3] "lu U", Solmazog, and H. Akisim, "A review on the elevator car suspension calculations," Elevator Technology 14: Proceedings of ELEVCON 2004, Istanbul, Turkey: IAEE Public, 2004.

- [4] Y.A. Onur, and CE. Imrak, "Computer aided car frame modelling and stress analysis," Elevator Technology 16: Proceedings of ELEVCON. Helsinki, Finland: IAEE Public, 2006.
- [5] FC. Babalik , K. Avdar, M. Sakalar, and B. Meshur , "Analysis and finite elements method aided design of elevator car suspensions," Asansor ANSYS, (55), 2003.
- [6] Y. Feng, J. Zhang, and Y. Zhao, "Modeling and robust control of horizontal vibrations for high-speed elevator," Journal of Vibration and Control, 15(9), 2009.
- [7] J.A.-K. Mohammed, F. M. Mohammed, and R. Ahmed, "Design and optimization of a surface-mounted permanent-magnet linear synchronous motor with Maxwell ANSYS", Al-Khwarizmi Engineering Journal 15(3), 2019.
- [8] R. Ahmed, F. M. Mohammed and J. Abdul-Kareem, "A single-side flat synchronous linear motor model," IOP Conf. Series: Materials Science and Engineering 765, 2020.
- [9] S. Othman, J. Abdul-Kareem and F. M. Mohammed, "Implementation of Linear Elevator with High Riding Quality Based on S-Curve Profile," The second international conference on electromechanical engineering and its applications (ICEMA-2021), Baghdad- Iraq, 2021.
- [10] P. Langer, M. Maeder, C. Guist, M. Krause and S. Marburg, "More Than Six Elements per Wavelength: The Practical Use of Structural Finite Element Models and Their Accuracy in Comparison with Experimental Results," Journal of Computational Acoustics, Vol. 25, 2017.
- [11] H. Igarashi, "Semi-Analytical Approach for Finite Element Analysis of Multi-turn Coil Considering Skin and Proximity Effects," IEEE Transactions on Magnetics, Vol. 53, Issue 1, pp. 1-8, 2017.
- [12] F. Al-Shamma, F. F. Mustafa and S. M. Saliman, "An Optimum Design of Cam Mechanisms with Roller Follower for Combined Effect of Impact and High Contact Loads," Al-Khwarizmi Engineering Journal, vol. 6, no. 4, 2010.
- [13] A. S. Jalal, N. J. Baker and D. Wu, "Electrical Machine Design for use in an External Combustion Free Piston Engine," 5th IET International Conference on Renewable Power Generation (RPG). London, UK, 2016.
- [14] Y. FU, "Foundation of Finite Element Analysis," Wuhan University Press, 2006.
- [15] S. P. Verma, "Design Optimization of 3hp, 4-Pole, 3-Phase, 50 Hz Induction Motor Employing Improved Genetic Algorithm," NIET Journal of Engineering & Technology, Vol. 1, Issue 1, 2012.