IJE TRANSACTIONS C: Aspects Vol. 35, No. 03, (March 2022) 572-579

International Journal of Engineering

Journal Homepage: www.ije.ir

Experimental Investigation and Numerical Simulation of Air Circulation in a Non-AC Bus Coach System

S. J. Niranjana[a]*, S. S. Kubsad[b], S. Manjunatha[c], Y. Nagaraj[d], I. Bhavi[e], B. M. Angadi[e], A. J. Chamkha[f], M. B. Vanarotti[g]

[a] *Department of Mechanical Engineering, School of Engineering and Technology, CHRIST (Deemed to be University), Bengaluru- 560076, Karnataka, India*
[b] *Department of Mechanical Engineering, Atria Institute of Technology, Bengaluru- 560024, Karnataka, India*
[c] *Department of Mathematics, School of Engineering and Technology, CHRIST (Deemed to be University), Bengaluru- 560076, Karnataka, India*
[d] *Department of Mechanical Engineering, SJM Institute Technology, Chitradurga-577502, Karnataka, India*
[e] *BLDEA's V.P. Dr. P.G. Halakatti College of Engineering and Technology, Vijayapur-586103, Karnataka, India*
[f] *Faculty of Engineering, Kuwait College of Science and Technology, Doha District, 7th Ring Road, 35004, Kuwait*
[g] *Sanjeevan Engineering & Technology Institute, Sanjeevan Knowledge City, Panhala, Kolhapur, Maharastra, India*

PAPER INFO

Paper history:
Received 29 November 2021
Received in revised form 21 December 2021
Accepted 23 December 2021

Keywords:
Air Circulation
CFD Analysis
Non-AC Bus
Wind Tunnel Test

ABSTRACT

Air circulation plays a vital role in the comfort of passengers in a bus, being a non-AC bus without any aid from the air conditioning system. The circulation of air is utterly dependent on the design of the bus and the natural flow of air. However, optimize the flow of air inside the bus, a study on the design of the bus is needed. In this regard, experimental work was carried out to achieve uniform airflow by redesigning the coach into an aerodynamic shape. The openings are provided at the leading edge of the bus to evaluate the best possibility for air to circulate in the bus. Three openings were provided at the leading edge of the bus, the first and second openings were mere openings, and the third opening was fitted with a roof vent providing three different geometric patterns to airflow. The initial boundary conditions were developed by considering that all windows and doors of the bus are closed. The scaling ratio of 1:20 was considered for modeling the bus. The experiments were conducted in the wind tunnel test rig. It was observed from the experimentation that the velocity of the air was considered to be the most influential parameter for the optimal air circulation. The velocities of 21.96 m/s and 22.68 m/s were obtained inside bus. The obtained experimental velocities were validated with results obtained by the Computational Fluid Dynamics (CFD). It was observed that a deviation of 5% for the given velocity of 20 m/s.

***doi**: 10.5829/ije.2022.35.03c.10*

NOMENCLATURE

v	Velocity of air (m/s)	g	Acceleration due to gravity (m/s^2)
ρ	Density (kg/m^3)	ρ_{air}	Density of air (kg/m^3)
Δh	Manometer height (m) (Final – Initial)		

1. INTRODUCTION

The non-AC bus is a major mode of transport in developing countries. During the journey passengers, comfort plays a vital role by Niranjan et al. [1]. The uneven airflow inside a non-AC bus may cause discomfort on the passenger. Kanekar et al. [2] stated the passenger's comfort will be improved by redesigning the existing structure of a bus. Drag force is the resistance offered by air against a moving object and which can be reduced with a small modification in the external geometry. Patidar et al. [3] experimented on an existing and modified bus model and proved a low drag coefficient. Norwazan et al. [4] states that reduction in

---

*Corresponding Author Institutional Email: *niranjana.s@christuniversity.in* (S. J. Niranjana)

Please cite this article as: S. J. Niranjana, S. S. Kubsad, S. Manjunatha, Y. Nagaraj, I. Bhavi, B. M. Angadi, A. J. Chamkha, M. B. Vanarotti, Experimental Investigation and Numerical Simulation of Air Circulation in a Non-AC Bus Coach System, ***International Journal of Engineering, Transactions C: Aspects*** Vol. 35, No. 03, (2022) 572-579

the fuel consumption and efficiency of the vehicle is determined by aerodynamic drag and coefficient of lift. Petzalla et al. [5] experimented on sedan-type vehicles with different vehicle geometrical features such as hood angle, rear angle, screen angle, and corner radius. They conclude the use of rear screen angle can reduce that drag. The crashes are happening due to the crosswind effect, a severe problem in some countries. The body's ideal shape suggests that a rounded front face, rounded. topsides, and sharp rear corners improve directional stability. Vollaro [6] stated the use of air screen doors and windows will improve the ventilation in non-AC buses during traffic signals and bus stops. During hot sunny days, the airflow inside a coach is an important aspect. Aliahmadipour et al. [7] explored the temperature and velocity distribution inside the coach. Heated manikins were used to study the temperature distribution with the passenger sitting and sleeping positions.

Shen et al. [8] found that the slight modification in the existing coach results in better passenger comfort. Rodrigues et al. [9] conducted an aerodynamic study and explored that changing small dimensions in the geometry reduces the drag and fuel consumption up to 20% and 10%, respectively.

The CFD analysis for a three-dimensional bus coach was carried out using ANSYS software, and the results are within the acceptable limit [10, 11]. Niranjana et al. [12] numerically analysed Ashok Leyland MTX Micra 28-seater on different types of boundary conditions and estimated the drag force would reduce to 0.67 by redesigning the bus structure.

Kale et al. [13] experimented on a non-AC bus, during the journey, the air enters from a rare window and moves towards the driver in a, and the velocity of air is almost one-tenth of the bus speed. They were opening the alternative window found to improve the air circulation in a non-AC-bus, increasing the passenger comfort [14, 15]. An experimental investigation was conducted on a scaled bus model in a wind tunnel test by placing the bus model exactly in the centre. The readings of drag and velocities were measured inside the bus [16, 17]. Niranjan et al. [18] provided a proper duct system that enhances passengers' comfort by the uniform airflow inside a non-AC bus. The amount of air required to cool the non-AC bus is the algebraic sum of heat generated from the glass, roof, passenger and engine loads are considered while designing a proper duct system and the cross-section of the duct by Fayazbakhsh and Bahrami [19]. Mathematical modelling of flow over surfaces was studied using CFD [20, 21]. Rahate et al. [22] designed an air distribution system for operation theatre using flow visualization techniques to improve flow characteristics. Kumar et al. [23] carried out an experimental study on flow characteristics around twin wind blades.

This work aims to create a uniform airflow in non-AC buses by providing adequate inlets near the front and top of the bus. The main issue in the existing public transportation system in developing and underdeveloped countries is achieving uniform airflow inside a non-AC bus. The bus will generally be completely filled during peak travel hours, with no available seats. The person in the window seat will experience a high flow of air and close the window to avoid it, causing the person standing in the aisle to suffocate.

The main objective of this work is to reach uniform airflow inside a non-AC bus system by redesigning the bus structure, modelling and fabricating a scaled bus model, and experimenting with and numerically simulating the scaled bus model.

2. MATERIALS AND METHODOLOGY

This work attempts to achieve uniform airflow inside the non-AC bus coach by providing proper openings at three levels, like two openings in the front and one in the bus's rooftop. 3-D model of passenger vehicle (bus) was developed by SOLIDWORK software as shown in Figure 1 with the dimensions of 350 mm in length and 105 mm in width and height, respectively. The topology of the fluid flow zone of interest was defined with the help of Computer-Aided Design (CAD) software. It was an essential aspect of the design and optimization process.

2. 1. Experimentation

A scaled MiTR non-AC bus model of 1:20 for experimental analysis is fabricated using a transparent acrylic sheet of thickness 6mm, as shown in Figure 2. Acrylic is a translucent synthetic material with excellent strength, stiffness and optical clarity. In contrast to many other transparent plastics, they possess superior weathering properties. Table 1 shows material property of acrylic sheet used to construct the bus model.

A scaled bus model is placed at the centre of the wind tunnel to measure the velocity. The provision is provided to measure the bus's velocity at three different positions, one is at the front of the first seat, the second in the middle

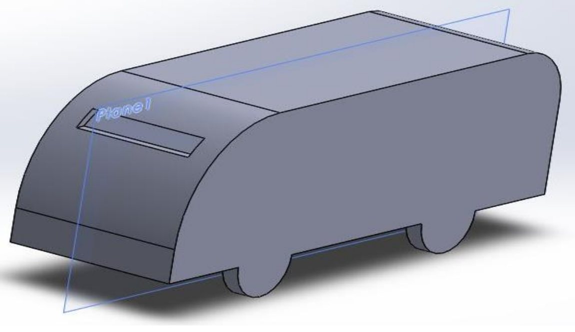
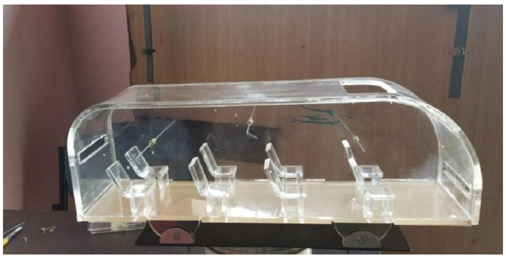
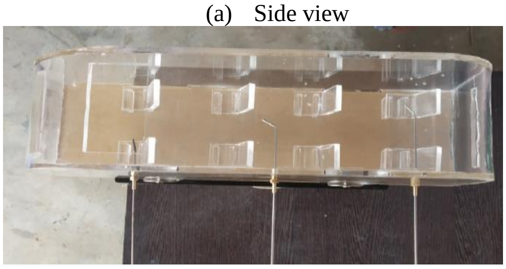


Figure 1. 1:20 Scaled model of non-AC bus



(a) Side view



(b) Top view

Figure 2. The scaled model of bus

TABLE 1. Material properties of acrylic sheet

Tensile Strength (MPa)	Flexural Modulus (MPa)	Light Transmission	Water Absorption	Density (g/cm^3)
75	115	>92%	0.2%	1.19

of the bus, and the last is provided in front of the last seat. The velocity port can be adjusted between the seats as shown in Figure 2b.

Wind tunnel test facility (600 x 600 x 2000) mm. is used for measuring the airflow inside a non-AC bus. The wind tunnel capacity of airflow is up to 45 m/s, and the setup has a facility for drag, lift, smoke and velocity tests, as shown in Figure 3. After repeated initial readings, the maximum error was noticed around 0.5%.

Recalibration and testing of wind gauges is done in wind tunnels. It usually contains a variety of settings for testing objects or recalibrating wind gauges in various wind speeds. The object or wind gauge will not be destroyed while in use, and the wind gauge will be able to measure the wind precisely and correctly both outdoors and indoors. This investigation concentrates on velocity distribution inside a non-AC bus. The test wind velocity can be set by adjusting the Pitot tube at the centre of the test area by using a lever mechanism. Initial readings in the static manometer are to be noted and then successive readings are noted to set the machine's RPM to conduct the experiment. The difference between initial and final readings will give a height. From Equation (1), the velocity at the inlet of the test chamber is calculated.

Figure 3. Wind tunnel test rig with scaled bus model

Once the required velocity of the machine is attained, and the machine's initial setup is done. The scaled bus model is placed exactly at the centre of the test chamber in a wind tunnel. The machine is run at 850 and 1015 RPM, respectively, to achieve the required velocity of 20m/s and 25m/s (considering the Indian driving cycle for ergonomic study for heavy passenger vehicles). Further, the manometer readings before starting the machine and later the successive readings at three different positions are noted. The initial and final reading difference gives the required height, and substituting in Equation (1), the velocity is calculated at the three different levels.

$$V = \sqrt{\frac{2\rho g \Delta h}{\rho_{air}}} \tag{1}$$

Initial readings in the multibank manometer are measured, and it found to be 19.4cm. The final readings at three different positions are measured and tabulated in Table 2.

2. 2. Boundary Conditions

The experimental analysis is carried for two different velocities through the inlet region for 20m/s and 25m/s, respectively. The Numerical analysis is carried out with a velocity of 20m/s. Only the inlets ports are kept open and all other openings (windows, doors) of the bus are kept closed.

TABLE 2. Multibank manometer readings for the five different cases

Case	RPM	Position 1 in cm	Position 2 in cm	Position 3 in cm
1	850	21.5	21.3	21.3
	1015	22.5	22.5	22.3
2	850	22.2	22.4	22.4
	1015	23.5	24	24
3	850	20.5	23.5	23.6
	1015	22	25	25.5
4	850	23.3	23.6	23.4
	1015	25	25.4	25.2
5	850	21.5	21.7	21.8
	1015	24.6	24.9	24.8

The computational domain surrounds the Inflow and outflow borders. Aside from symmetry and solid walls, it is assumed that the velocity distribution is uniform at the inflow and pressure is assumed to be zero-gradient at the outflow. Table 3 indicates the assumed conditions and equations for the numerical analysis. There are four parameters for the convergence pressure, temperature, momentum and turbulence. 10-6 for K- ε and 10-4 for pressure, velocity and temperature.

3. RESULTS AND DISCUSSION

3. 1. Experimental Investigation

This section discusses the detailed results of a successful trial of wind tunnel test for airflow in the non-AC bus. In the first trial, the machine is kept at the height of 7.9 cm and run at 700rpm speed, and later machine ends with the same speed with a height of 9.5 cm. Substituting these readings in Equation (1), we noticed that velocity is 16 m/s. But the required velocity to conduct a test on the scaled bus model is 20m/s and 25m/s. To achieve the target, the speed of the machine is increased from 700 to 800rpm. Thus, obtained velocity is found to be 19.2 m/s. After several successive trials, it is noticed that if the machine runs at 850rpm and 1015rpm, the velocity is found to be 20m/s and 25m/s, respectively.

After setting the standard experimental set up the specimen is placed at the chamber's centre. Figure 4 represents the position of slots at three different test cases, respectively. In case 1, the openings are provided only in the first slot of the front end. And all other regions are closed. In case 2, the openings provided only on the roof vent outside, and all other slots are closed. In case 3, the openings provided only in the top slot and all other slots are closed. Case 4 illustrates the outlets provided only inside the roof vent and all other slots closed. In case 5, the openings are provided only in the second slot of the frontend and all other slots are closed. The value of the velocity of the air flow in the bus for all five different cases is tabulated in Table 4.

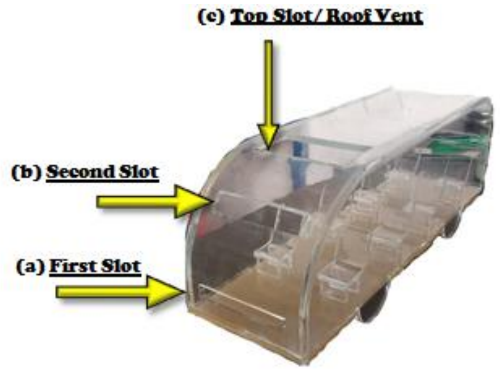


Figure 4. Scaled bus model with different test cases

It is found that out of all five cases result, the case five results show that the air's velocity is uniform in the non-AC bus coach.

Based on the experimental results, Figures 5a to 5e are plotted to explain the input and output velocity at three different positions.

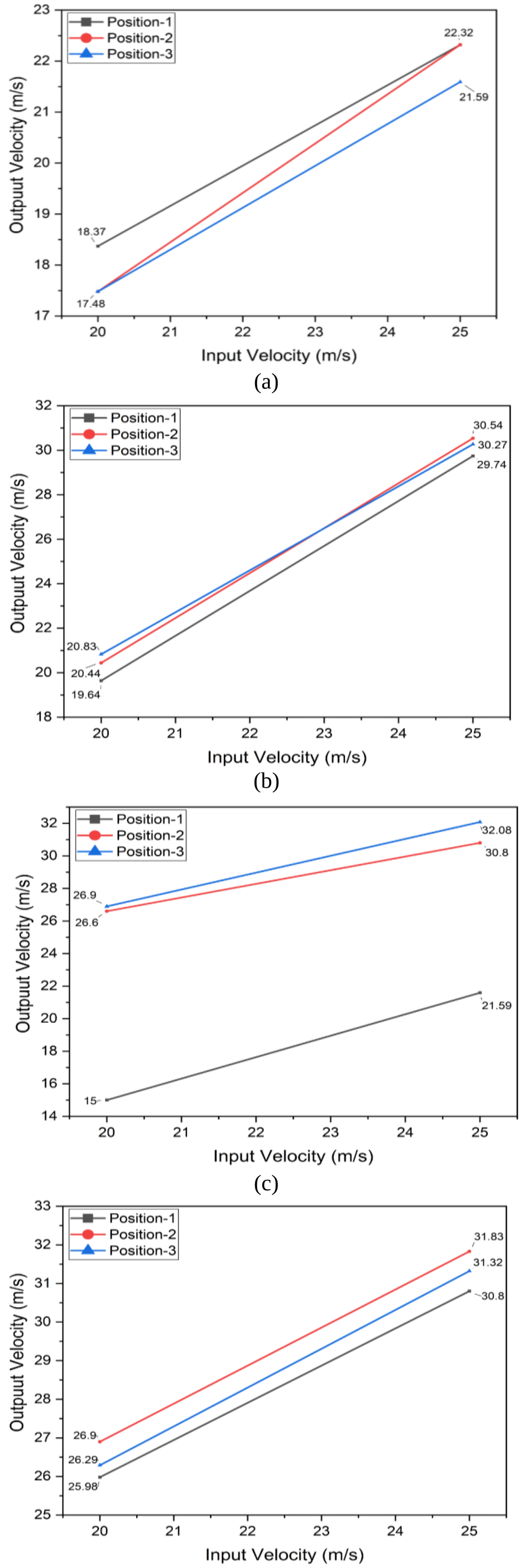
From Figures 5a and 5b, it is observed that velocity is not uniform. Since air entered in Figure 5a is hitting the passenger's foot, air will be diverted non-uniformly, and from Figures 5b and 5c, it is observed that the air entered from the roof vent will not pass to the driver and few passengers sitting next to the driver. Hence the uniform air circulation is not found. Figures 5d and 5e explore that air velocity is found to be uniform in all three positions, which is due to air entering from the front end and in the

TABLE 3. Appendix

Type	Explanation
Governing Equation	Navier stoke equation • Pressure-momentum • Mass conservation • Continuity viscosity • K- ε equation $\rho\frac{Du}{Dt}=\rho\left(\frac{\partial u}{\partial t}+u.\nabla u\right)=$ $\nabla \mathrm{p}+\nabla\left\{\mu\left(\nabla \mathrm{u}+(\nabla \mathrm{u})^{T}-\frac{2}{3}(\nabla.\mathrm{u})\mathrm{I}\right)+\right.$ $\left.\xi(\nabla.\mathrm{u})I\right\}+\rho g$
Initial Condition	It is running at atmospheric pressure condition ie normal temperature and pressure (NTP).
Boundary Condition	At inlet velocity, outlet- zero pressure and walls – Adiabatic
Assumptions	The velocity near the entire setup remains constant. There is no variation in the property of fluid and temperature.

TABLE 4. Velocity of air at different position

Cases	RPM	Input Velocity in m/s	Velocity at Position 1 in m/s	Velocity at Position 2 in m/s	Velocity at Position 3 in m/s
Case 1	850	20	18.37	17.48	17.48
	1015	25	22.32	22.32	21.59
Case 2	850	20	15	26.6	26.9
	1015	25	21.59	30.8	32.08
Case 3	850	20	25.98	26.9	26.29
	1015	25	30.8	31.83	31.32
Case 4	850	20	19.64	20.44	20.83
	1015	25	29.74	30.54	30.27
Case 5	850	20	21.96	22.68	22.68
	1015	25	26.3	27.78	27.78

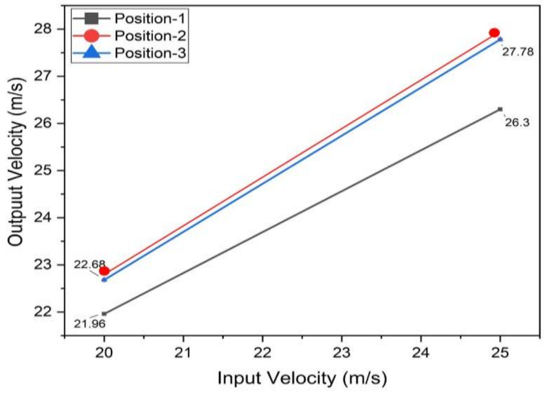


(a)

(b)

(c)

(d)



(e)

Figure 5. Velocity distribution curve

appropriate height of the sitting position of passengers. Hence, we can conclude that obstacles for airflow are comparatively less in case of Figure 5e.

3. 2. CFD Analysis

The CFD analysis for case 5 (opening in the front and all other regions are closed) is considered, as this case results are better among the different test cases. The discretization of a scaled non-AC bus model is as shown in Figure 6.

The three-dimensional tetra mesh and 779151 elements are used for the analysis. The quality parameters like aspect ratio, Jacobian, minimum and maximum angle, skew angle and warpage are maintained during the discretization process. Assign an inlet, outlet, body, side and top as different layers in the bus, and these files are imported on a three-dimensional layout with the ANSYS Fluent CFD code in a staggered grid system.

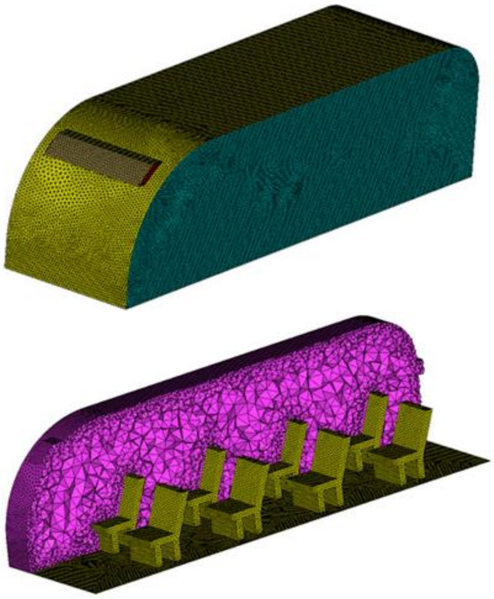
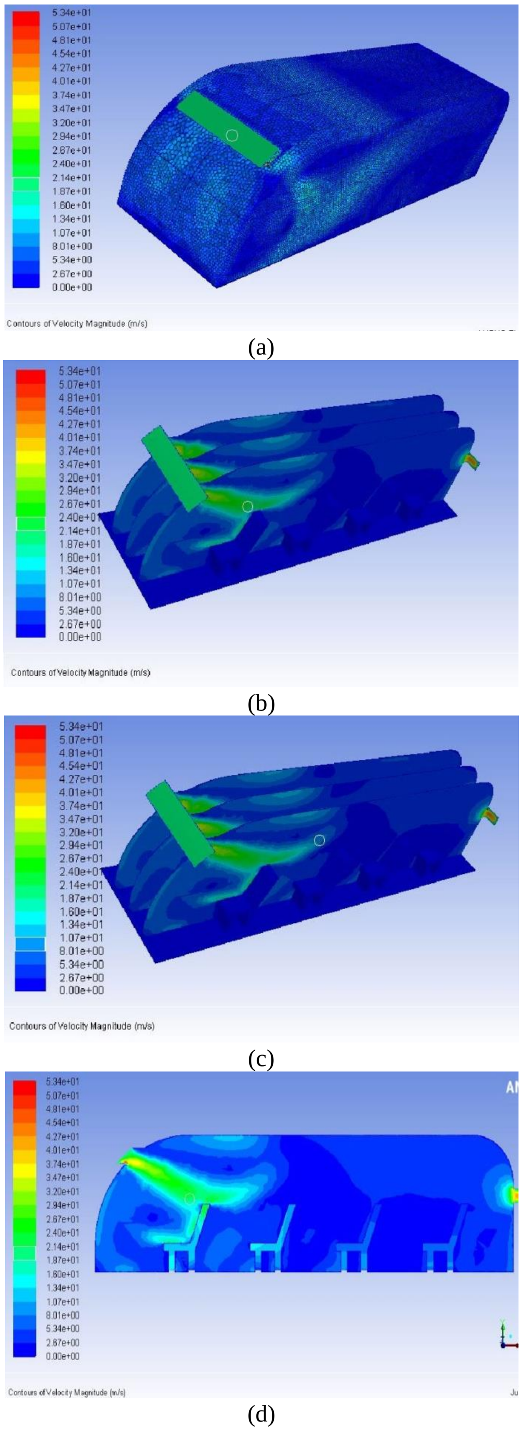


Figure 6. Meshed model of the bus

The governing equations are solved using the finite-volume approach, and the turbulence effect is analyzed with an equation k-ε model.

The CFD analysis is carried out using ANSYS Fluent 16.0 software, and velocity is measured at different regions. The input velocity of 20m/s is assigned in the front slot of the bus, as shown in Figure 7a. Figure 7b explores that air velocity is 21m/s in front of first row chairs of the bus, and the velocity is 15m/s in the middle of the aisle, as shown in Figure 7c. Finally, it is found that the outlet velocity is measured to be 22m/s, as depicted in Figure 7d. The recirculation of air near the inlet and outlet results in vertex formation, which increases the velocity slightly, as shown in Figures 7b and 7d.



(a)

(b)

(c)

(d)

Figure 7. CFD results using ANSYS fluent software

Table 5 represents the comparison results of experimental and CFD analysis. It is found that the velocity at the front and rear end match with CFD analysis results, and velocity at the middle of the aisle is comparatively less, which is due to obstacles in the bus.

Figure 8 represents the experimental and numerical method results. The velocity at three different positions is shown.

Table 6 represents the number of iterations conducted for the CFD analysis. As the number of elements is increased, the velocity at position one inside the bus is decreased, and at some iterations, the value of velocity is closely matching hence convergence is achieved. The values are plotted in Figure 9. In order to reduce the computational time for the numerical analysis 779151 elements were considered for the study.

TABLE 5. Comparison results between experimental and numerical methods

Input Velocity of 20 m/s	Experimental Results in m/s	Numerical Results in m/s
In front of first seat	21.96	21
Middle of aisle	22.68	15
Near last seat	22.68	22

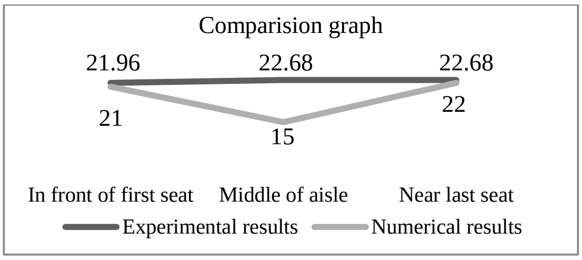


Figure 8. CFD and experimental results comparison

TABLE 6: Analysis iteration

Sl No	Number of Elements	Velocity at Position 1 in m/s
1	10000	41.6
2	100000	35.2
3	300000	29.4
4	500000	24.6
5	790000	21
6	900000	20.8
7	110000	21.04

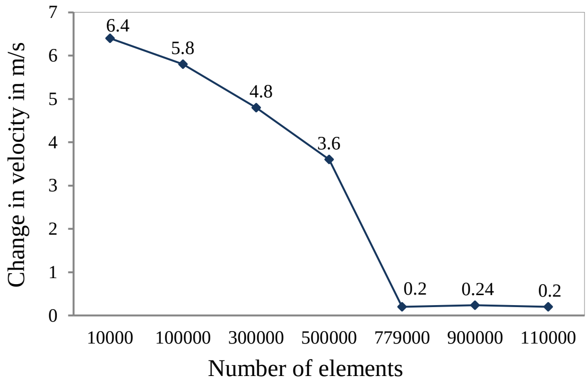


Figure 9. Mesh independency (Convergence criteria)

4. CONCLUSION

An experimental investigation of uniform airflow for a scaled MiTR non-AC bus model of 1:20 has been conducted. The bus model is fabricated using transparent acrylic sheet of thickness 6mm. The numerical simulation of uniform airflow for the test model is performed using CFD. The major outcome of the study reveals that the velocity distribution curve is uniform in the case 5 test condition and the numerical and experimental results are in close agreement with a deviation of less than 5%.

5. REFERENCES

1. Niranjana, S. J., Kubsad, S. S., Ravichandran, G., and Santhosh, N. "A Comprehensive Understanding of Airflow in Non Air conditioned Bus Coaching System." ***International Journal of Engineering and Advanced Technology***, Vol. 9, No. 3, (2020), 1222–1225. https://doi.org/10.35940/ijeat.C5317.029320
2. Kanekar, S., Thakre, P., and Rajkumar, E. "Aerodynamic study of state transport bus using computational fluid dynamics." ***IOP Conference Series: Materials Science and Engineering***, Vol. 263, (2017), 062052. https://doi.org/10.1088/1757-899X/263/6/062052
3. Patidar, A., Gupta, U., and Bansal, A. "Fuel Efficiency Improvement of Commercial Vehicle by Investigating Drag Resistance." In SAE International, 2015. https://doi.org/10.4271/2015-01-2893
4. Norwazan, A. R., Khalid, A. J., Zulkiffli, A. K., Nadia, O., and Fuad, M. N. "Experimental and Numerical Analysis of Lift and Drag Force of Sedan Car Spoiler." ***Applied Mechanics and Materials***, Vol. 165, (2012), 43–47. https://doi.org/10.4028/www.scientific.net/AMM.165.43
5. Petzäll, J., Torlund, P. Å., Falkmer, T., Albertsson, P., and Björnstig, U. "Aerodynamic design of high-sided coaches to reduce cross-wind sensitivity, based on wind tunnel tests." ***International Journal of Crashworthiness***, Vol. 13, No. 2, (2008), 185–194. https://doi.org/10.1080/13588260701788476
6. Vollaro, R. de L. "Indoor Climate Analysis for Urban Mobility Buses: a CFD Model for the Evaluation of thermal Comfort." ***International Journal of Environmental Protection and Policy***, Vol. 1, No. 1, (2013), 1. https://doi.org/10.11648/j.ijepp.20130101.11
7. Aliahmadipour, M., Abdolzadeh, M., and Lari, K. "Air flow simulation of HVAC system in compartment of a passenger coach." ***Applied Thermal Engineering***, Vol. 123, (2017), 973–990. https://doi.org/10.1016/j.applthermaleng.2017.05.086
8. Shen, K., Li, F., and Ni, J. "1D/3D Coupling Calculation Analysis on Bus Cooling System." ***Energy and Power Engineering***, Vol. 06, No. 14, (2014), 550–556. https://doi.org/10.4236/epe.2014.614048
9. Rodrigues, A. F. A., Gertz, L. C., Cervieri, A., and Telh, M. A. "Aerodynamic Analysis of a Vehicle Minibus." In SAE Technical Paper Series, 23rd SAE Brasil International Congress and Display, 2014. https://doi.org/10.4271/2014-36-0327
10. Shafie, N. E. A., Kamar, H. M., and Kamsah, N. "Effects of air supply diffusers and air return grilles layout on contaminants concentration in bus passenger compartment." ***International Journal of Automotive Technology***, Vol. 17, No. 5, (2016), 751–762. https://doi.org/10.1007/s12239-016-0074-1
11. Thabet, S., and Thabit, T. H. "CFD Simulation of the Air Flow around a Car Model (Ahmed Body)." ***International Journal of Scientific and Research Publications (IJSRP)***, Vol. 8, No. 7, (2018), 2250–3153. https://doi.org/10.29322/IJSRP.8.7.2018.p7979
12. Shravanabelagola Jinachandra, N., Sadashivappa Kubsad, S., Sarpabhushana, M., Siddaramaiah, S., and Rajashekaraiah, T. "Modeling and computational fluid dynamic analysis on a non-AC bus coach system." ***Heat Transfer***, Vol. 49, No. 8, (2020), 4870–4877. https://doi.org/10.1002/htj.21857
13. Kale, S. R., Veeravalli, S. V., Punekar, H. D., and Yelmule, M. M. "Air flow through a non-airconditioned bus with open windows." ***Sadhana***, Vol. 32, No. 4, (2007), 347–363. https://doi.org/10.1007/s12046-007-0029-3
14. John, P., Sriram, B., R, S. K., Kumar, S. V., Ramasamy, P., and Vijay Ram, C. "Ventilation Improvement in a Non-AC Bus." In SAE International, 2013. https://doi.org/10.4271/2013-01-2457
15. S, V., C, V. R., and P, S. "Evaluation of Bus Ventilation Methods Using CFD." In ***SAE International***. https://doi.org/10.4271/2013-26-0043
16. Ünal, Ş. "An Experimental Study on a Bus Air Conditioner to Determine its Conformity to Design and Comfort Conditions." ***Journal of Thermal Engineering***, Vol. 3, No. 1, (2016), 1089–1089. https://doi.org/10.18186/thermal.277288
17. Moureh, J., Menia, N., and Flick, D. "Numerical and experimental study of airflow in a typical refrigerated truck configuration loaded with pallets." ***Computers and Electronics in Agriculture***, Vol. 34, No. 1–3, (2002), 25–42. https://doi.org/10.1016/S0168-1699(01)00178-8
18. Niranjana, S., Kubsad, S. S., Nagaraj, Y., Manjunath, S., and Shivakumar, S. "Experimental Investigation of Air Circulation Using Duct System in a Non-AC Bus Coach." ***IOP Conference Series: Materials Science and Engineering***, Vol. 1065, No. 1, (2021), 012006. https://doi.org/10.1088/1757-899X/1065/1/012006
19. Fayazbakhsh, M. A., and Bahrami, M. "Comprehensive Modeling of Vehicle Air Conditioning Loads Using Heat Balance Method." In SAE International, 2013. https://doi.org/10.4271/2013-01-1507
20. Askari, S., and Shojaeefard, M. H. "Mathematical Modeling of Potential Flow over a Rotating Cylinder." ***International Journal of Engineering, Transactions A: Basics***, Vol. 24, No. 1, (2011), 55–63. Retrieved from https://www.sid.ir/en/Journal/ViewPaper.aspx?ID=194565
21. Sharifipour, M., Bonakdari, H., and Zaji, A. H. "Impact of the Confluence Angle on Flow Field and Flowmeter Accuracy in Open Channel Junctions." ***International Journal of Engineering, Transactions B: Applications***, Vol. 28, No. 8, (2015), 1145–1153. Retrieved from

https://www.ije.ir/article_72560.html

22. Rahate, S. D., and Sarode, A. D. “Design of Air Distribution System for Operation Theatre Using Flow Visualization Techniques to Improve Flow Characteristics.” ***International Journal of Engineering, Transactions A: Basics***, Vol. 33, No. 1, (2020), 164–169. https://doi.org/10.5829/IJE.2020.33.01A.19

23. Kumar, R., and Saranprabhu, M. K. “Experimental Study on Flow Characteristics Around Twin Wind Blades (RESEARCH NOTE).” ***International Journal of Engineering, Transactions B: Applications***, Vol. 31, No. 5, (2018), 820–825. https://doi.org/10.5829/ije.2018.31.05b.18

Persian Abstract

چکیده

گردش هوا نقشی حیاتی در راحتی مسافران اتوبوس ایفا می‌کند، زیرا یک اتوبوس بدون AC بدون هیچ کمکی از سیستم تهویه مطبوع بهره‌مند است. گردش هوا کاملاً به طراحی اتوبوس و جریان طبیعی هوا بستگی دارد. با این حال، برای بهینه‌سازی جریان هوا در داخل اتوبوس، مطالعه در مورد طراحی اتوبوس مورد نیاز است. در این راستا، کار آزمایشی برای دستیابی به جریان هوای یکنواخت با طراحی مجدد اتوبوس به شکل آیرودینامیکی انجام شد. دهانه‌ها در لبه جلویی اتوبوس برای ارزیابی بهترین امکان برای گردش هوا در اتوبوس فراهم شده است. سه دهانه در لبه جلویی اتوبوس تعبیه شده بود، دهانه اول و دوم بازشونده بودند و دهانه سوم با یک دریچه سقفی تعبیه شده بود که سه الگوی هندسی مختلف را برای جریان هوا فراهم می‌کرد. شرایط مرزی اولیه با در نظر گرفتن بسته بودن تمام پنجره‌ها و درهای اتوبوس ایجاد شد. نسبت مقیاس ۱:۲۰ برای مدل سازی اتوبوس در نظر گرفته شد. آزمایش‌ها در دکل آزمایشی تونل باد انجام شد. از آزمایش مشاهده شد که سرعت هوا به عنوان تأثیرگذارترین پارامتر برای گردش بهینه هوا در نظر گرفته شد. سرعت های ۲۱/۹۶ متر بر ثانیه و ۲۲/۶۸ متر بر ثانیه در داخل اتوبوس به دست آمد. سرعت‌های تجربی به‌دست‌آمده با نتایج به‌دست‌آمده توسط دینامیک سیالات محاسباتی (CFD) تأیید شد. انحراف ۵ درصد برای سرعت داده شده ۲۰ متر بر ثانیه مشاهده گردید.