PAPER • OPEN ACCESS

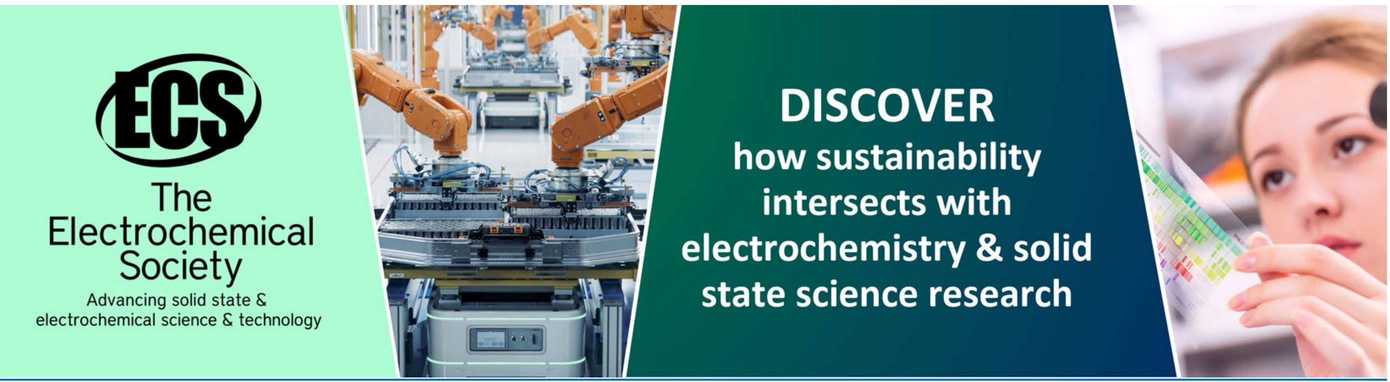
CFD Investigation on Long-Haul Passenger Bus

To cite this article: C F Tan *et al* 2015 *IOP Conf. Ser.: Mater. Sci. Eng.* **88** 012025

View the article online for updates and enhancements.

You may also like

- Carbon dioxide emissions from international air transport of people and freight: New Zealand as a case study
Anna P Tarr, Inga J Smith and Craig J Rodger
- Coupling simulation of the cooling air duct and the battery pack in battery energy storage systems
Xinlong Zhu, Xintian Xu, Benben Kong et al.
- A tutorial on fiber Kerr nonlinearity effect and its compensation in optical communication systems
Sunish Kumar Orappanpara Soman



This content was downloaded from IP address 3.139.81.210 on 14/05/2024 at 21:14

CFD Investigation on Long-Haul Passenger Bus

C F Tan[1], B T Tee[2], H C Law[3], T L Lim[4]
[1,2,3] Centre for Advanced Research on Energy (CARe)
Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka,
Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia

[4] TKH Manufacturing Sdn Bhd
Kota Damansara, Selangor, Malaysia

E-mail: cheefai@utem.edu.my

Abstract. Air flow distribution is one of the important factors that will influence the bus passenger comfort during long haul travel. Poor air flow distribution not only cause discomfort to the bus passenger but also influence their travel mode as well. The main purpose of this study is to investigate the air flow performance of the bus air-conditioning system through CFD simulation approach. A 3D CAD model of air ducts was drawn and hence analysed by using CFD software, namely ANSYS Fluent, to determine the airflow rate for every outlets of the air-conditioning system. The simulated result was then validated with experimental data obtained from prototype model of air duct. Based on the findings, new design concepts is proposed with the aim to meet the industry requirement as well as to improve the bus passenger comfort during long haul travel.

1. Introduction

Automotive air-conditioning (AAC) for thermal comfort in passenger cabins is now a thing of necessity rather than luxury and cooling is especially needed when travelling in summer or throughout the year in countries of hot and humid climate [1]. Application of automotive air-conditioning in the field of passenger transportation was necessary in order to ensure the comfort of passenger and driver while travelling long distance. Thus, control of airflow rate and temperature distribution of automotive air-conditioning system is essential to ensure optimal humidity and comfort inside a long-haul passenger bus. This study emerged based on a need to further investigate the current air flow distribution system in the long haul express bus that build by the Malaysian local coach manufacturer. The express bus is facing the issue of uneven airflow rate when the air is delivered from the air-conditioning system. Therefore, a Computational Fluid Dynamics (CFD) study is necessary in order to investigate the air flow pattern and air flow distribution of the express bus air conditioning ducting, which will help to identify the locations where air distribution is inadequate.

[1] To whom any correspondence should be addressed.

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Published under licence by IOP Publishing Ltd

2. Methodology

2.1 Parameter Description

The current air ducts that fitted to the Hino RK-1J SLL long-haul passenger bus have been studied. The overall dimension for the front portion of air ducts at the bus driver's side was measured by using a measuring tape. A 3D CAD modelling of the air ducts with outlets was drawn by using CATIA according to the measured dimensions. The airflow rate from the air ducts outlets was measured by using integral vane anemometer. After that, analysis will be carry out by running simulation with defined parameters on air ducts CAD model with the aid of ANSYS Fluent. Figure 1 showed the outlook of Hino RK1J-SLL long-haul passenger bus.

Figure 1. Outlook of Hino RK1J-SLL long-haul passenger bus.

The overall air ducts dimension was obtained through measurement by using measuring tape at Pioneer Coachbuilders Sdn. Bhd. Furthermore, location and the size of the outlets were measured in order to construct an accurate 3D modelling of air ducts inside CATIA. Figure 2 showed the actual outlook of air ducts while Figure 3 and 4 shown the round outlets and long outlets respectively that fitted to the air ducts.

Figure 2. Air ducts outlook.

Figure 3. Round air ducts outlets.

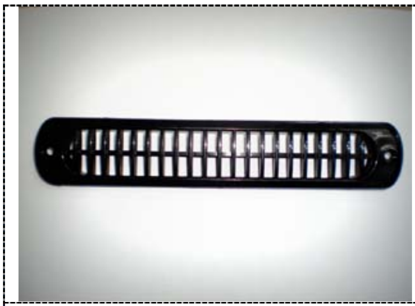


Figure 4. Long air ducts outlet.

2.2 Measurement Procedure

The rotating vane anemometer (RVA) which had been used widely in various industries has proved useful in air flow measurement. This device used a lightweight rotating vane as the sensor to measure the airflow rate [2]. Airflow rate from the air ducts outlets was measured by using an AZ Instrument 8903 integral vane anemometer. The measurement took place inside of Hino RK1J-SLL long haul passenger bus. The bus was at stationary during the measurement and the data was taken every interval of 30 minutes. The location of outlets was shown in Figure 5.

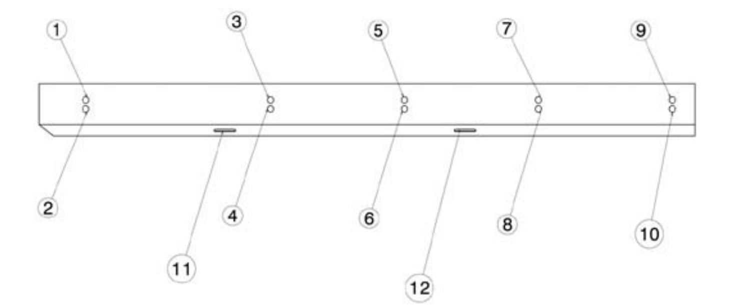


Figure 5. Location of outlets (bottom view).

2.2 Simulation Procedures

The excessive edge of the actual air ducts was eliminated and the outlets was simplified by eliminated the details of vane structure. Hence, a simplified 3D CAD model of air ducts with inlet and multiple outlets was constructed inside CATIA according to the measured dimension. The 3D CAD model of air ducts was as shown in Figure 6 below.

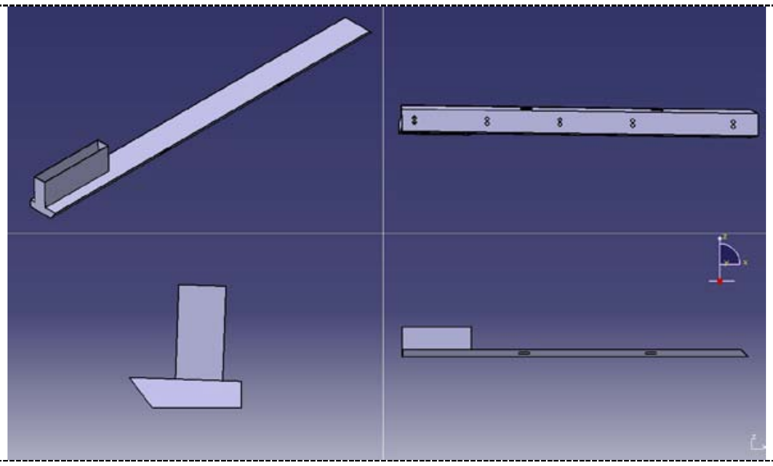


Figure 6. 3D CAD modelling of air duct.

The CFD analysis begins with meshing of the internal volume of air ducts inside ANSYS Design Modeller environment. Hence the meshed model was simulated inside ANSYS Fluent environment with defined boundary conditions. Fluent having all the physical capabilities needed to model flow, turbulence, heat transfer and reactions for industrial applications. Finite volume approach was used in Fluent to solve the equations governing the flow and is widely used in numerical simulations of different flow conditions of various complexities [3]. It is chosen for the simulation because its proven capability and validity in flows. The simulated results will then compared to the results obtained from experimental results.

The internal volume of air ducts was meshed inside ANSYS Design Modeller environment by using the automatic meshing function. The following meshing parameter was defined:

a) Relevance centre = Fine
b) Smoothing = High
c) Inflation = Program Controlled
d) Boundary Layer = 10 Layers

The other meshing parameter was remain default and the meshed model generated having 60253 of total elements.

3. Results and Discussions

The simulation results were obtained after the simulation comes to converged, that the solution no longer changes with subsequent iterations. Overall mass, momentum, energy, and scalar balances are achieved after converged. The velocity magnitude (m/s) for the internal volume of air ducts was shown in Figure 7.

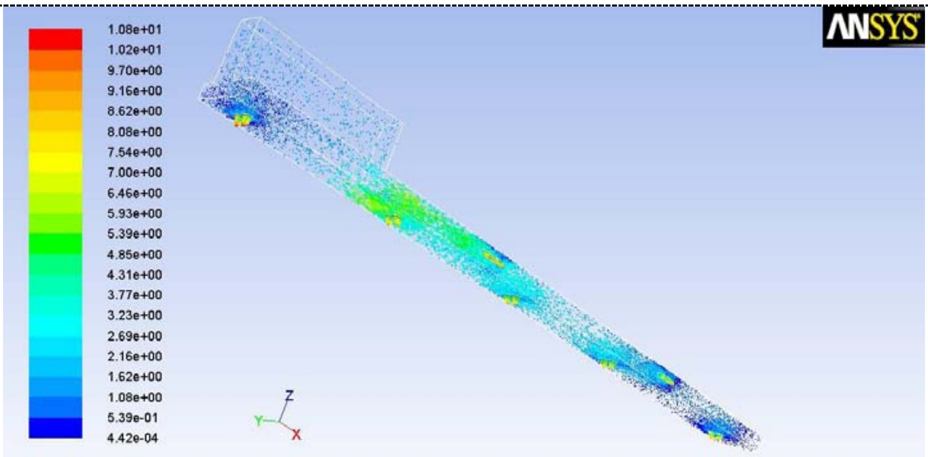


Figure 7. Velocity magnitude (m/s) for the internal volume of air ducts.

The simulation results that simulated under ideal condition usually different from the experimental results that obtained under actual operation [4]. The experimental and simulation results were compared in order to obtain the percentage of difference between experimental and simulation results. The data of experimental results and simulation results were tabulated in Table 1. Based on the table, it is shown that there is a high percentage of difference between experimental and simulation results for the outlets. The highest percentage of difference was 79.22% for outlet 2 while the lowest was 39.62% for outlet 9. This difference occurred as the simulation was run at ideal condition as the wall friction of the air ducts was not considered during the simulation.

Table 1. Comparison of experimental and simulation results.

Outlet	Experimental (m/s)	Simulation (m/s)	% of Difference
1	5.64	8.89	57.62
2	5.10	9.14	79.22
3	5.80	9.08	56.55
4	5.39	9.29	72.36
5	6.23	9.30	52.49
6	5.83	9.19	57.63
7	5.67	9.32	64.37
8	5.67	9.11	60.67
9	7.42	10.36	39.62
10	7.21	10.32	43.13
11	6.26	9.03	44.25
12	5.92	9.76	64.86

Besides that, the complex vane structure for the outlets was eliminated in the simulation, which the air will flow freely through the outlets without any resistance from the vane structure. Furthermore, the inlet flow was assumed to be constant during the simulation, that is not applicable to the actual air conditioning unit as the air conditioning unit that supply the airflow not always running at highest efficiency.

Despite the big difference between the experimental and simulation results, the simulation results showed that the outlet 1 having the lowest airflow rate of 8.89 m/s. This indicated that the outlets that located on bus driver's top having lower airflow rate compare to other outlets for both experimental and simulation results.

Design modification to the air ducts later was conducted in order to obtain a desired solution to improve the airflow rate. The internal volume of new air ducts design with deflector was constructed inside CATIA. The deflector was place at the behind the flow path of outlet 1 and outlet 2 in order to redirect more airflow to outlet 1 and outlet 2. Besides, the deflector also served the purpose to prevent the air flow to the end of air ducts while ensure the airflow directly exit through outlet 1 and outlet 2. Heuristic approach was used to determine the most effective angle of deflector, thus internal volume of air ducts with deflector at 90°, 75°, 60° and 45° was constructed. The CFD simulation of airflow rate for outlet 1 and 2 with deflector at 90° was shown in Figure 8 and 9 respectively.

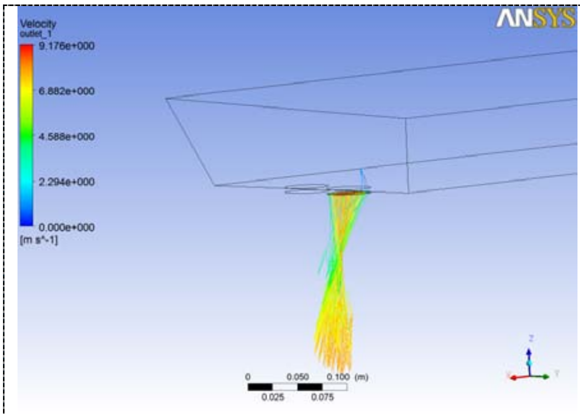


Figure 8. Airflow rate of outlet 1 for deflector 90°.

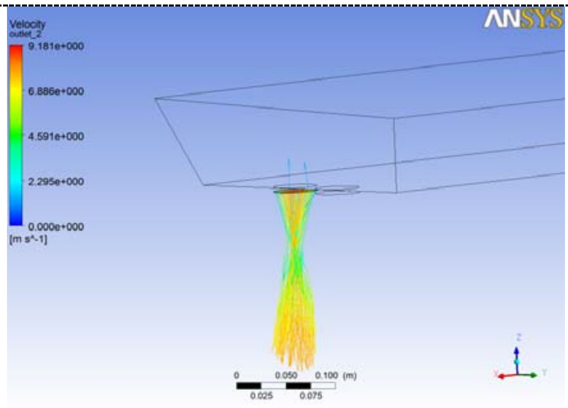


Figure 9. Airflow rate of outlet 2 for deflector at 90°.

The simulation results for original air ducts and new air ducts design with deflector at 90°, 75°, 60° and 45° was compared in Table 2 for outlet 1 and Table 3 for outlet 2. The percentage of improvement was determined by setting the simulation results of original air ducts as datum. From Table 2, it shown that the air flow rate for outlet 1 of new air ducts design with deflector at 60° having significant improvement of 6.52% over the original air ducts design. It poses higher percentage of improvement compare to 3.26% for deflector at 90, 4.50% for deflector at 75° and 4.72% for deflector at 45°.

Table 2. Percentage of improvement for outlet 1.

Degree of Deflector (°)	Datum (m/s)	Airflow Rate of Outlet 1 (m/s)	% of Improvement
90	8.89	9.18	3.26
75	8.89	9.29	4.50
60	8.89	9.47	6.52
45	8.89	9.31	4.72

Meanwhile from Table 3, it shown that the air flow rate for outlet 2 of new air ducts design with deflector at 75° showed the highest improvement of 2.63% over the original air ducts design. Its percentage of improvement followed by 2.41% for deflector at 60, 1.20% for deflector at 45° and 0.44% for deflector at 90°.

Table 3. Percentage of improvement for outlet 2.

Degree of Deflector (°)	Datum (m/s)	Airflow Rate of Outlet 2 (m/s)	% of Improvement
90	9.14	9.18	0.44
75	9.14	9.38	2.63
60	9.14	9.36	2.41
45	9.14	9.25	1.20

4. Conclusions

CFD analysis had been carried out to study the airflow rate of original air ducts and new air ducts design with deflector. The study focussed on percentage of improvement for the airflow rate at outlets along bus air-conditioning system. The objective of this study was achieved as the airflow rate delivered from outlets of air conditioning air ducts for long-haul passenger bus was improved with new design of air ducts. The new design of air ducts was fitted with deflector to redirect the airflow and the deflector was found to be effective at the angle of 60° through heuristic approach.

References

[1] Haslinda M K Mohd Y S & Nazri K 2012 Computerized Simulation of Automotive Air-Conditioning System: Development of Mathematical Model and Its Validation *International Journal of Computer Science* **9(2)** 2
[2] Burgess W A Ellenbecker M J & Treitman R D 2004 *Ventilation for Control of The Work Environment* 2nd ed Hoboken (New Jersey: John Wiley & Sons Inc)
[3] Fluent 2011 *Fluent 14.0 User's Guide* (Fluent Inc: USA)
[4] Shiu H R Ou F C & Chen S L 2002 3C Duct Design Method *The Chinese Journal of Mechanics – Series A* **18(2)**