International Journal of Engineering & Technology, 7 (4.13) (2018) 188-194

International Journal of Engineering & Technology

Website: www.sciencepubco.com/index.php/IJET

Research paper

CFD simulation of local exhaust ventilation for aviation fire-test lab

Aiman S H Al-Ammari[1], Yazan S M Altarazi[1], Abd. Rahim Abu Talib[1,2,]*, M I Nadiir Bheekhun[3]

[1]*Aerodynamics, Heat Transfer and Propulsion (AHTP) Group, Department of Aerospace Engineering, Faculty of Engineering, Universiti Putra Malaysia, 43400 Serdang, Selangor, Malaysia.*
[2]*Aerospace Malaysia Research Centre (AMRC), Faculty of Engineering, Universiti Putra Malaysia, 43400 Serdang, Selangor, Malaysia.*
[3]*Aerospace Science and Technology Research Group, Faculty of Information Sciences and Engineering, Management and Science University, 40100 Shah Alam, Selangor*
**Corresponding author E-mail: abdrahim@upm.edu.my*

Abstract

Excellent indoor air quality in an enclosed area has always become a major safety aspect in designing a building. Issues with regards to circulation of air and exhaust system must be first resolved before the said building can be used for any purposes. The goal of this study is to identify ways to improve air quality in the aviation fire test room at the Propulsion Laboratory that is located in Universiti Putra Malaysia (UPM), Selangor, Malaysia. A computational fluid dynamics (CFD) method was employed to predict the air contaminant inside the lab. When performing the activities, the indoor air quality have to be ensure circulated and ventilated in the lab. Using a mechanical fans and natural ventilation are a traditional method to provide indoor air quality into the propulsion. Whereby, this method may not be enough to provide the required indoor air quality for specific aviation fire-test setup. Such labs may suffer from increasing air contaminant based on the improper and irregular air distribution. A grid independent test (GIT) was done to reduce the effects of meshing on the results was carried out to estimate the discretization error. Computational fluid dynamic (CFD) method was carried out to identify a suitable ventilation system that would result in the greatest improvement in the indoor air quality (IAQ) inside the lab. The results of using the CFD simulation show that installing Local Exhaust Ventilation (LEV) at the lab could significantly improve the IAQ inside the lab. The airflow increase by 84% and the CO, CO_2 and NO reduce by 84%, 89 and 81%, respectively. Improvement of the IAQ by increasing the airflow and reducing in the air CO, CO_2, and NO, which can be considered as very significant achievement.

Keywords: *ventilation; CFD; simulation; local exhaust ventilation; indoor air quality.*

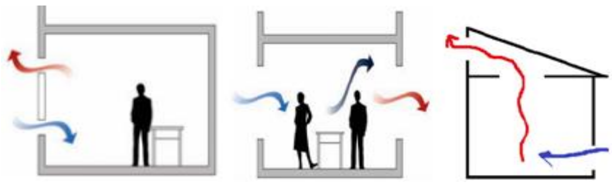
1. Introduction

One of the major environmental concerns is indoor air quality (IAQ) since people almost between their workplace and their residence, they spend about 7% of their daily time commuting and about 90% of their time indoors [1]. Over the past several decades has been seen the IAQ as a major issue. Nosocomial infections threaten human health, due to a poor combination of air quality and practical flow. A ventilation systems that ensures the health of the workers and avoiding unnecessary risks, when a professional activity involves exposure to dust, gas or any types of air contaminant.

Ventilation system is the most method used in many buildings for improve indoor air quality. The ventilation is defined as the process by which fresh air is introduced and the removal of ventilated air from an occupied space [2]. The primary aim of ventilation is to preserve the qualities of air. Sometimes, ventilation may also be used to lower the temperature inside an occupied area. Several types of ventilation can be used to control the air distribution and to provide an indoor air quality in buildings such as natural, mechanical, and hybrid ventilation [3]. The method bypasses the outside air to a space through openings doors, windows, and using natural forces to ventilation [4]. A method using mechanical devices such as exhaust vents and fans to supply and remove air called mechanical ventilation [5]. A process using combine ventilation systems as a mechanical ventilation and natural ventilation to provide the indoor air quality called hybrid ventilation [2, 6]. The natural ventilation is recommended as a typical system at the humid and hot climate regions [7, 8]. Whereby, it is not capable of providing sufficient level of indoor air quality in all areas, due to the inconsistent wind speed and different climate characteristic.

Many labs building in Malaysia supply an indoor air quality by using method such as a mechanical fans and natural ventilation. The low wind velocity with an average of 1.5 m/s was provided during the year, that data recorded by the Malaysian Meteorological service for 10 years period [9]. The mechanical fans only move the air inside the space but they do not promote exchange of fresh air [10-17]. Hence, the presently used ventilation system is ineffective to provide a satisfactory level of indoor air quality [18]. Therefore, an alternative ventilation strategy is needed. A lab is considered as important buildings in Malaysia. It is a place for the people to perform their some experiments such as experiment nozzle and turbine units, air compressor unit and accessories and axial flow gas turbine engine. When performing the occupants' activities, the indoor air quality have to be ensured flowing into the lab. However, there is a lack of in-depth study and analysis of indoor air quality into lab buildings. There are several tools and methods that can be used to study and analyze ventilation system in buildings. These include empirical models, analytical models, zonal models, multi-zone models, small-scale experimental models, full-scale experimental models, and computational fluid dynamics (CFD) [19]. The computational fluid dynamics (CFD) method is convenient, accurate and widely used in predicting the ventilation performance. The rapid increase in computing capability has made this method even more popular [19-21]. A combination of CFD analysis and field measurement has been used in many studies to assess the indoor air quality in buildings [22-28].

In this study, a similar approach is employed to evaluate the performance of a proposed ventilation system for a lab building. Indoor air quality (IAQ) is a term which refers to the air quality within and around buildings and structures, especially as it relates to the health and comfort of building occupants. Reducing the risk of indoor health concerns by controlling and understanding the common pollutants indoors [29]. NO is nitrogen monoxide and part of a group of gaseous air pollutants. They are produced during combustion, especially at high temperature [30]. The standards and guidelines of indoor air environment is a basic reference for contaminants concentration level. The National Institute for Occupational Safety and Health (NIOSH), Occupational Safety and Health Administration (OSHA), Environment Protection Agency (EPA) and World Health Organization (WHO) are the organizations which have developed the standards and guidelines for the indoor environment [31-33]. Introducing and removing of ventilated air from an occupied space and replace the fresh air instead of it this process called ventilation [2]. There are two purposes to ventilate the occupied spaces in building which are: the first goal is providing a lowering the air temperature and a heat transport mechanism into an occupied space to provide thermal comfort. The second aim is removing the indoor pollutions and supplying fresh air to provide an acceptable indoor air quality (IAQ) [19]. Three elements are used to determine the performance of ventilation systems in buildings. These are air change rate (ACH), airflow direction and air distribution [36]. Ventilations using natural and displace the air space bypass specified openings as windows or doors this process called natural ventilation [4]. The types of natural ventilation illustrated into three parts [4, 5].



(a) (b) (c)

Fig. 1: natural ventilation divided to three main types; (a) Single-sided Ventilation (b) Cross-Ventilation, and (c) Stack Ventilation

Predicting the thermal comfort and ventilation performance using the accurate, popular, and widely employed which is CFD [19, 20]. It is utilized to simulate the parameters of thermal comfort, for example, air temperature, airflow velocity and relative humidity [36, 37]. In addition, CFD method also is used to assess the thermal comfort in buildings with reduced time, cost and human resource [19]. CFD method also allows users to efficiently take full control over the boundary conditions and easily perform parametric studies [28, 38]. Throughout past studies, it was found that in Malaysian climatic condition, natural ventilation and mechanical fans are commonly used to provide cooling in many buildings, including labs.

In Malaysia, natural ventilation and mechanical fans are commonly used to provide indoor air quality in many buildings, including labs. This paper is aimed to identify ways to improve air quality in propulsion lab that; located in Universiti Putra Malaysia (UPM), Selangor, Malaysia. Section 2 is focused on the methodology which used to achieve the solution for this study which include a development of the CFD Model in section 2.1, GIT in section 2.2, section 2.3 discussed about boundary conditions and properties, for section 2.4 include solver, solution methods and convergence and CFD simulation on the effects of an (LEV) in section 2.5. In additional, section 3 is focused on the results which include CFD simulation of this study and existing case, comparison of (LEV) with existing case and air flow pattern in section 3.1, 3.2, 3.3, and 3.4, respectively. Finally, the summary of CFD results in section 4. This paper is intended to use the CFD simulation to show that by installing Local Exhaust Ventilation (LEV) at the lab could significantly improve the IAQ inside the lab.

2. Methodology

2.1. Development of the CFD model

As a case study, the lab that located in Universiti Putra Malaysia (UPM), Selangor, Malaysia was selected for this research. The photo of the lab is shown in Figure 2. Isometric view for a simplified model of the lab with its dimensions are illustrated in Figure 3. The lab has an overall size of length, L = 6 m, width, W = 4 m and height, H = 4 m and volume of the lab considered is 96 m^3. The two doors have the dimensions of 1.2 m and 2 wide and 2 m height.

Fig. 2: Photo of the lab

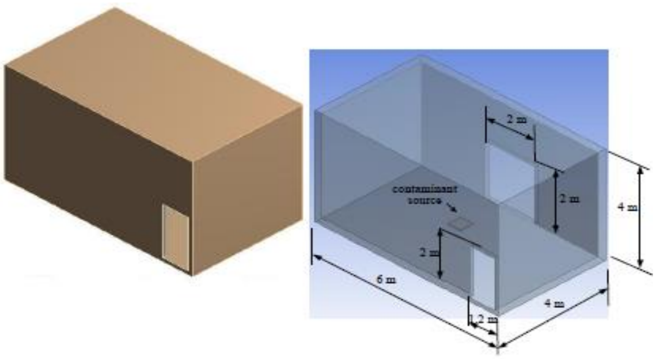


Fig. 3: Simplified CFD model of the lab

2.2. Grid Independence Test (GIT)

Qualitative grid verification was performed by GIT using different grid sizes to determine the effect of the grid size on the results [21]. The GIT was started from coarse (default) size to finer size, elements size of 1,072,752 and 6,227,052 respectively. The finest five sets of meshing result are shown in Figure 4. As the figure illustrates, increasing the number of elements more than 2,798,544 gives similar air contaminant values. This indicates that the number of elements at 2,798,544 can be considered as adequate in eliminating the effects of meshing on the results. Therefore, it was used throughout the CFD simulations.

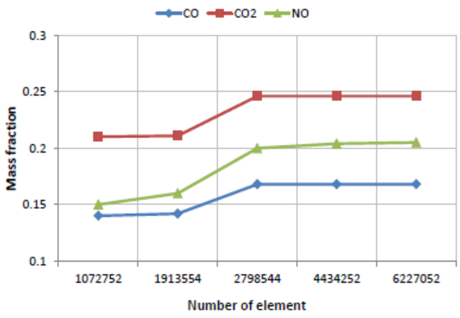


Fig. 4: Effect of grid size on predicted air contaminant

Figure 5(a) illustrated the computational domain for meshing. The grid in areas where there large gradients in the flow vibration was refined. The cell size of the critical part normally smaller by 10 times than the cell size of the surrounding critical zone. Meanwhile, the cell size of the surrounding critical zone should be smaller by 2.5 times than the cell size of the whole geometry [40]. In this study, the surrounding critical zone were divided as follows, the coarser cells were designed for the entire geometry, fine cells were created for the high sensitivity zone (material, doors, and windows), medium cells size for the surrounding sensitivity zone is shown in Figure 5(b).

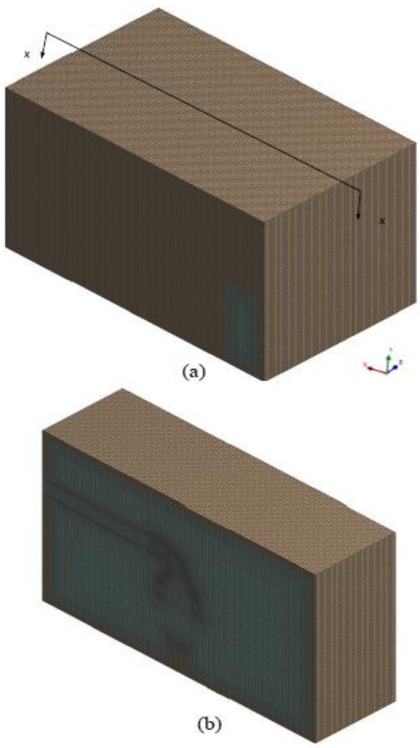


Fig. 5: (a) Meshing of the computational domain, and (b) A view of section x-x

The grids are good or bad quality has to check by the mesh matric. The mesh values in particular skewness is <0.95 and orthogonal quality is <0.1 are recommended [41]. The orthogonal quality in the CFD model is 0.97 and skewness is 0.13 of the grid elements. The ratio between size of adjacent cells, minimum cell size and maximum cell size used in the CFD model are 1:2, 0.002 m and 0.253 m, respectively. The mesh metric of the 2,798,544 elements and the properties are illustrated in the Table 1, which indicate a good mesh quality.

Table 1: Properties of the mesh

Size		Mesh metric	
Maximum	0.253 m	Skewness (close to 0.0)	0.13
Minimum	0.002 m	Orthogonal	0.97
Curvature	18°	Element quality	0.98
Growth rate	1:2		

2.3. Boundary conditions and properties

In this stage, boundary conditions for the computational domain of the building were specified. The actual field measurement was provided the boundary condition parameters, which are: air contaminants, airflow and wall temperature. All boundary conditions prescribed on the CFD model are shown in Table 2. The properties of the air, air contaminants and concrete were obtained from the literature and they are presented in Table 3 and Table 4.

Table 2: Boundary conditions used in CFD model

Section		Boundary condition	Parameters
Right door		Inlet air	V_a, door = 0.5 m/s T_a = 28.2°C Species mass fraction: Air (O_2: 0.23, N_2: 0.76, H_2O: 0.01
Left door		Outlet air	Pressure = 0 Pa (gauge)
Wall	Roof	Temperature	T_w,r = 34.4°C
	Front		T_w, front = 28.8°C
	Rear		T_w, rear = 29.5°C
	Right		T_w,R = 29°C
	Left		T_w,l = 29.2°C
	Floor		T_w,F = 28°C
Occupants		Temperature	37°C
		Metabolic Rate	150 W
		Heat generation rate	15 W/m^3

Table 3: Properties of air and concrete [34]

Properties	Air	Concrete
Density (m^3/kg)	1.134	2400
Thermal conductivity (W/m.K)	0.0262	0.9
Specific heat (J/kg.K)	1006.4	0.75
Molecular weight (kg/kg.mol)	28.69	-
Specific gas constant (J/kg.K)	287.1	-
Viscosity (kg/m s)	1.86321 × 10^{-5}	-

Table 4: Properties of gas [34]

		Gas	
Properties	CO	CO_2	NO
Density (m^3/kg)	1.1233	1.787	1
Specific heat (J/kg.K)	0.025	0.0145	0.0454
Molecular weight (kg/kg.mol)	28	44	30

2.4. Solver, solution methods and convergence

Reaching the converged solution by using the much lower residuals was recommended from validation studies [42]. The residual absolute values for all equations are set to 10^{-4} except the energy where the value is set to 10^{-6} [39, 41] for this study. This is to ensure that the residual equations are complete converged. The residual absolute is used for the steady-state cases. The convergence of the iterations of the simulations of CO, CO_2 and NO are shown in Figure 6.

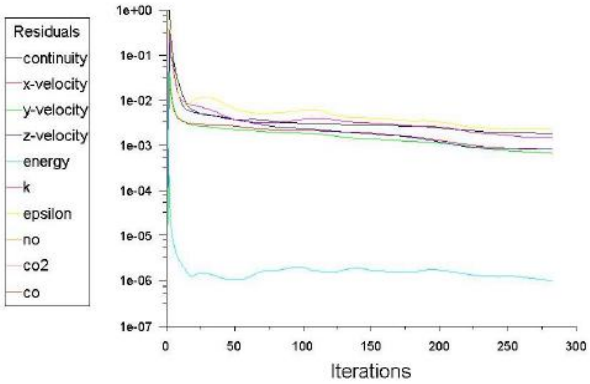


Fig. 6: Convergence graph of existing case in steady-state

2.5. CFD simulation on the effects of a local exhaust ventilation (LEV)

The CFD method used to investigate the influence of the Local Exhaust Ventilation (LEV) on IAQ inside the lab. It is expected that appropriate ventilation system helps in providing a more uniform and controlled airflow conditions and reduced the air contaminant in the lab, which in turns improve the IAQ. In this paper, the ductwork system consists of straight duct, fittings, flow control devices (exhaust fan), hood and supports. The length of straight duct equals 2.5 m with rectangular hood with 1 m width and 1 m depth and used fittings are elbow to change the gas stream direction, typically by 45°. The exhaust fan was placed at the end of the straight duct. The exhaust fan at the front will pull the air contaminant and hot air out of the space causing fresh air to be drawn into space through any available openings as shown in Figure 7. The LEV was modeled as are shown in Figure 7. The previous boundary conditions which have mentioned before are used the same boundary in the CFD model of the LEV. In addition, boundary conditions of the LEV are shown in Table 5.

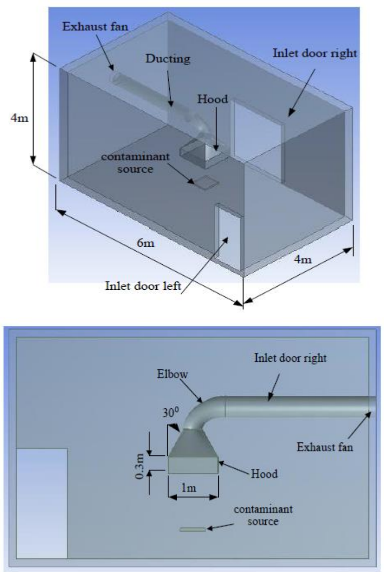


Fig. 7: Simplified CFD model of the lab with Local Exhaust Ventilation (LEV)

Table 5: LEV settings

Properties	Value
Ducting length	2.5 m
Hood size	1 m width and 1 m depth
Elbow	40°
Mass flow rate of exhaust fan	0.363 kg/s
Temperature	300 K
Diameter	0.4 m
Upper limit absolute pressure	101340 pa
Lower limit absolute pressure	101315 Pa

The important factors that affect the IAQ have to compare are airflow and air contaminant. The comparison has been done for the performances of the existing ventilation system and the LEV.

3. Results and discussion

3.1. CFD simulation

The CFD simulation results for all cases of airflow and air contaminant are presented in the form of contour plots on the sections plan A-A, as shown in Figure 8(a). The airflow and air contaminant also are presented in the form of a graph on the line X-X. The line marked were taken at the width and pass in the middle of the lab. The sampling locations of airflow and air contaminant are shown in Figure 8(b). There are three locations namely North (P_1), Middle (P_2), South (P_3). These locations were taken in 0.7 m height from the ground level.

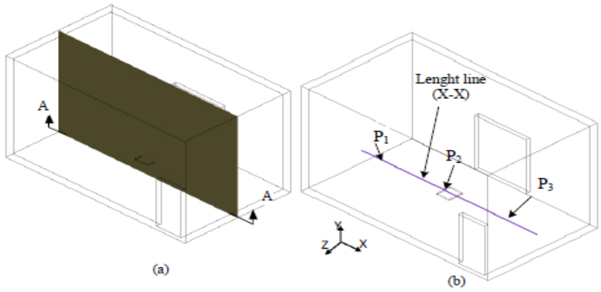
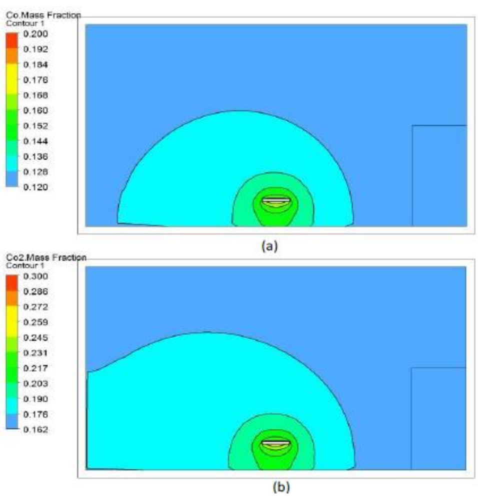
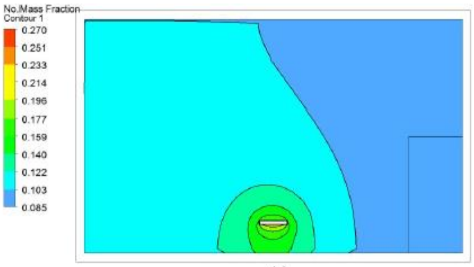


Fig. 8: (a) Locations of cross sections A-A plan, (b) Sampling locations and reference line of airflow and air contaminant

3.2. CFD simulation of the existing case

The existing case represents the natural ventilation system for the lab. Figure 9, shows the contour plot of CO, CO_2 and NO contaminant distribution for the existing case at steady–state condition. It can be seen that, the air contaminant distribution inside the lab is not uniform. The CO ranges from 0.12 to 0.2 on the section plane A-A, are shown in Figure 9(a). The figure also indicates that the CO in the zone close to the combustion material is about 0.2 and it decreases gradually away from combustion material towards the middle region of lab. It can be also observed from this figure that the CO_2 ranges from 0.162 to 0.30 and NO ranges from 0.085 to 0.270 on the same section plane A-A are shown in Figure 9 (b and c). Additionally, the figure clearly shows that the CO_2 and NO around the combustion material are the highest and it decreases when moving away from the combustion material. This is because the combustion material releases CO, CO_2 and NO from their material to the surrounding air of lab.



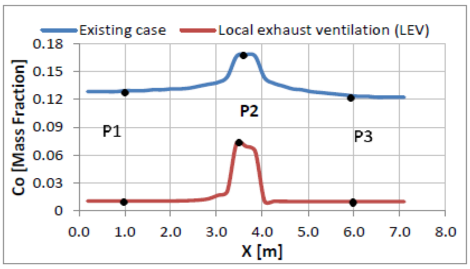


(c)

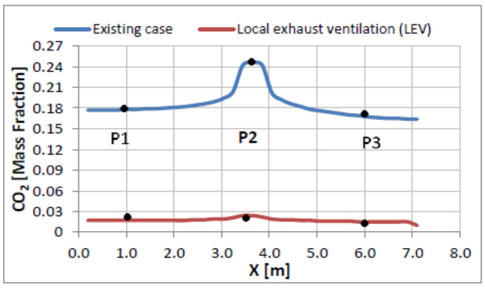
Fig. 9: Existing case: Air contaminant distribution (a) CO (b) CO_2, and (c) NO

3.3. Comparison of the LEV with existing case

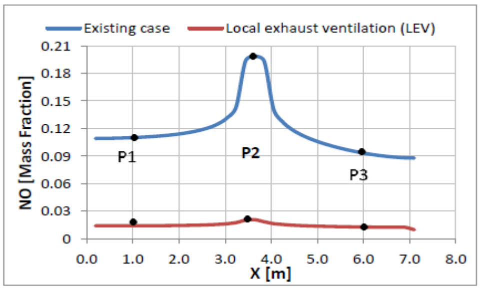
Figure 10 and Figure 11 illustrate the comparison graph between the existing case and the LEV for a variation airflow and air contaminant. The variation of CO, CO_2 and NO along the line Z-Z are shown in Figure 10. The CO variation inside the lab for the two cases is shown in Figure 10(a). It can be seen that the CO at the location P1 is reduced from 0.13 to 0.02. This shows that the CO has dropped by 0.11, which is equivalent to 84% CO reduction. The CO at the P_2 and P_3 locations has dropped by 0.095 and 0.1, which is equivalent to 54% and 83%, respectively. Figure 11(b) shows the plots of CO_2 for the LEV and the existing case. It can be seen that the CO_2 at location P_1 reduced from 0.18 to 0.02, which is equivalent to 89% reduction compared to the existing case. Whereas the CO_2 at the P_2 and P_3 locations reduced by 0.22 and 0.16, which is equivalent to 91% and 88%, respectively, compared to the existing case. Figure 10(c) is shown a two cases for variation NO into the lab. The location P1 is seen that the NO has been dropped from 0.1 to 0.02, which is equivalent to 81% compared to the existing case. The NO at the P_2 and P_3 has dropped by 0.17 and 0.1, respectively. This is equivalent to a reduction of 85%and 81%, respectively. The corresponding comparative plots of the CO, CO_2 and NO values of the LEV with baseline case are shown in Table 6. Figure 11 shows the plots of airflow for LEV installing at the lab and that for the existing case. It can be seen that the airflow at location P_1 increases from 0.08 m/s to 0.5 m/s, which is equivalent to 84% increment compared to the existing case. Whereas the airflow at the P_2 and P_3 locations increases by 0.43 m/s and 0.4 m/s, which is equivalent to 78% and 80%, respectively, compared to the existing case.



(a)



(b)



(c)

Fig. 10: Comparison of the air contaminant variation along width line X-X, (a) CO, (b) CO_2, and (c) NO contaminant between existing case and LEV

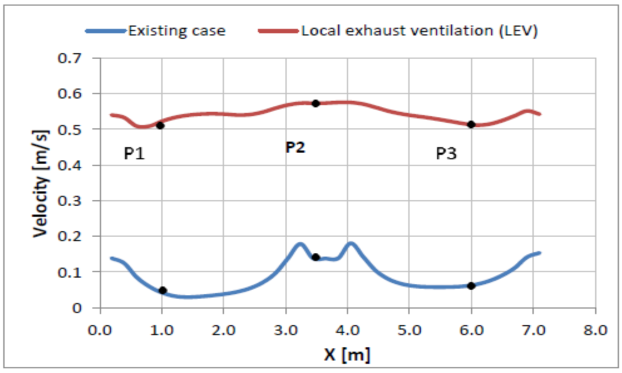


Fig. 11: Comparison of the air flow variation along width line X-X between existing case and LEV

In summary, the LEV installation at the lab has a potential to reduce both the air contaminant of CO, CO_2 and NO, and increase the airflow inside the lab room. It also produces a more uniform air contaminant and airflow distribution inside the lab compared to the existing case.

Table 6: Comparison of the air contaminant between existing case and Local exhaust ventilation (LEV)

		Baseline case	LEV	Reduction (%)
CO	P_1	0.13	0.02	84%
	P_2	0.16	0.07	54%
	P_3	0.12	0.01	83%
CO_2	P_1	0.18	0.02	89%
	P_2	0.25	0.03	91%
	P_3	0.17	0.02	88%
NO	P_1	0.11	0.02	81%
	P_2	0.20	0.02	85%
	P_3	0.10	0.01	81%

3.4. Pattern of the air flow

Figure 12 are shown the distribution of airflow into the lab for the existing case and LEV. The airflow for the existing case from the east door towards the loophole on the west-wall of the lab is shown in Figure 12(a). The airflow forth at the center segment of the lab on the grounds that the temperature is higher in this section due to the heat generated from the combustion material. Figure 12(c) shows the airflow distribution inside the lab when LEV was installed at the lab. The airflow at the middle area while it is passing from the door to the exhaust fans as the Figure 12 shown. A uniform distribute of the airflow insure anywhere of the lab, which in turn helps to exhaust the air contaminant from the inside to outside lab room. This situation reduces the air contaminant in all regions inside the lab.

5. Conclusion

Results of CFD flow simulations show that installing LEV in the lab could significantly improve the IAQ inside the lab. The airflow increases by 84% and the CO, CO_2 and NO reduces by 84%, 89% and 81%, respectively. The increase in the airflow and the reduction in the CO, CO_2 and NO improve the IAQ. Furthermore, LEV installation at the lab promotes a sufficient air flow inside the lab, through sucking of the air contaminant from the envelope, allowing fresh air to be circulated through the lab. The advantages of LEV were enhancing the air circulation, distribution and hence removed the air contaminant in the lab, leading to improving the IAQ in all regions inside the lab.

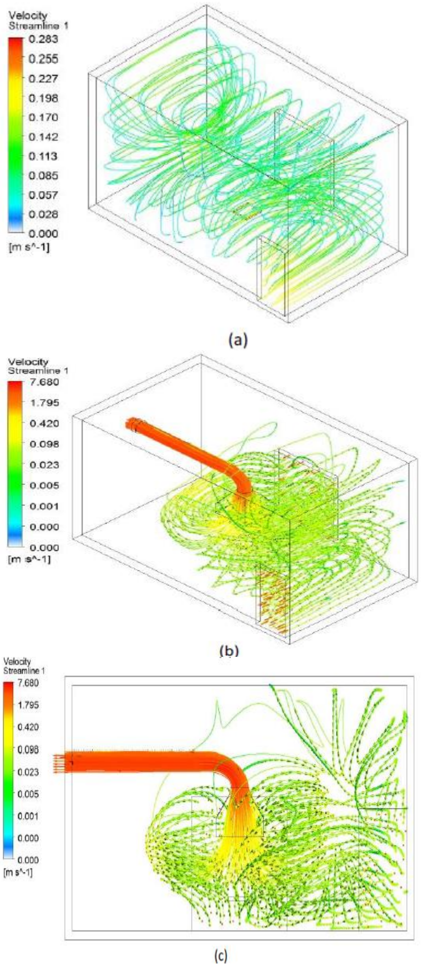


Fig. 12: Airflow distribution inside the lab; (a) Existing case, (b) LEV and (c) A view of section LEV

Acknowledgment

The authors would like to thank Mr. Mohd Nasir Johari, Assistant Engineer at the Propulsion Laboratory, Faculty of Engineering, UPM for his assistance. The study is part of the Master of Innovation and Engineering Design project at UPM. This work is supported by the Ministry of Higher Education Malaysia under the Fundamental Research Grant Scheme FRGS/TK05/UPM/02/7 no. 5524896.

References

[1] Klepeis NE, Nelson WC, Ott WR, Robinson JP, Tsang AM, Switzer P, Behar JV, Hern SC & Engelmann WH (2001), The National Human Activity Pattern Survey (NHAPS): a resource for assessing exposure to environmental pollutants. *Journal of Exposure Science and Environmental Epidemiology* 11(3), 231

[2] Heiselberg P, Murakami S & Roulet C (1998), Ventilation of large spaces in buildings. Final Report IEA Annex

[3] Cao G, Awbi H, Yao R, Fan Y, Sirén K, Kosonen R & Zhang JJ (2014), A review of the performance of different ventilation and airflow distribution systems in buildings. *Building and Environment* 73, 171-186

[4] Elzaidabi AAM (2009). *Low energy, wind catcher assisted indirect-evaporative cooling system for building applications*. University of Nottingham

[5] Abdullah AH (2007), *A study on thermal environment performance in atria in the tropics with special reference to Malaysia*. Herriot-Watt University

[6] Wouters P, Heijmans N, Delmotte C & Vandaele L (1999), Classification of hybrid ventilation concepts. IEA Annex.

[7] Kubota T, Chyee DTH & Ahmad S (2009), The effects of night ventilation technique on indoor thermal environment for residential buildings in hot-humid climate of Malaysia. *Energy and Buildings* 41(8), 829-839

[8] Rahman AMA, Khalid HL & Yusup Y (2011), Optimizing wind power for energy efficient building design in tropical hot-humid climate of Malaysia. *Journal of Sustainable Development* 4(2), 217

[9] Hussein I, Rahman M & Maria T (2009), Field studies on thermal comfort of air-conditioned and non air-conditioned buildings in Malaysia. 3rd International Conference on Energy and Environment

[10] Raja I, Nicol J, McCartney K & Humphreys M (2001), Thermal comfort: use of controls in naturally ventilated buildings. *Energy and Buildings* 33(3), 235-244

[11] Zhai Y, Zhang H, Zhang Y, Pasut W, Arens E & Meng Q (2013), Comfort under personally controlled air movement in warm and humid environments. *Building and Environment* 65, 109-117

[12] Wong NH, Song J & Istiadji A (2006), A study of the effectiveness of mechanical ventilation systems of a hawker center in Singapore using CFD simulations. *Building and Environment* 41(6), 726-733

[13] Lee S & Chang M (2000), Indoor and outdoor air quality investigation at schools in Hong Kong. *Chemosphere* 41, 109-113

[14] Khedari J, Yamtraipat N, Pratintong N & Hirunlabh J (2000), Thailand ventilation comfort chart. *Energy and Buildings* 32(3), 245-249

[15] Atthajariyakul S & Lertsatittanakorn C (2008), Small fan assisted air conditioner for thermal comfort and energy saving in Thailand. *Energy Conversion and Management* 49(10), 2499-2504

[16] Sekhar S (2005), Space temperature difference, cooling coil and fan—energy and indoor air quality issues revisited. *Energy and Buildings* 37(1), 49-54

[17] Al-Homoud MS, Abdou AA & Budaiwi IM (2009), Assessment of monitored energy use and thermal comfort conditions in mosques in hot-humid climates. *Energy and Buildings* 41(6), 607-614

[18] Chow W, Fung W & Wong L (2002), Preliminary studies on a new method for assessing ventilation in large spaces. *Building and Environment* 37(2), 145-152

[19] Chen Q (2009), Ventilation performance prediction for buildings: A method overview and recent applications. *Building and Environment* 44(4), 848-858

[20] Hurnik M, Blaszczok M & Popiolek Z (2015), Air distribution measurement in a room with a sidewall jet: A 3D benchmark test for CFD validation. *Building and Environment* 93(2), 319-330

[21] van Hooff T, Blocken B, Aanen L & Bronsema B (2011), A venturi-shaped roof for wind-induced natural ventilation of buildings: wind tunnel and CFD evaluation of different design configurations. *Building and Environment* 46(9), 1797-1807

[22] Coffey C & Hunt G (2007), Ventilation effectiveness measures based on heat removal: Part 2. Application to natural ventilation flows. *Building and environment* 42(6), 2249-2262

[23] Fitzgerald SD & Woods AW (2008), The influence of stacks on flow patterns and stratification associated with natural ventilation. *Building and Environment* 43(10), 1719-1733

[24] Ji Y, Cook M & Hanby V (2007), CFD modelling of natural displacement ventilation in an enclosure connected to an atrium. *Building and Environment* 42(3), 1158-1172

[25] Juárez JO, Hinojosa JF, Xamán JP & Tello MP (2011), Numerical study of natural convection in an open cavity considering temperature-dependent fluid properties. *International Journal of Thermal Sciences* 50(11), 2184-2197

[26] Rajagopalan P & Luther MB (2013), Thermal and ventilation performance of a naturally ventilated sports hall within an aquatic centre. *Energy and Buildings* 58, 111-122

[27] Hajdukiewicz M, Geron M & Keane MM (2013), Calibrated CFD simulation to evaluate thermal comfort in a highly-glazed naturally ventilated room. *Building and Environment* 70, 73-89

[28] Gilani S, Montazeri H & Blocken B (2013), CFD simulation of temperature stratification for a building space: validation and sensitivity analysis. 13th Conference of International Building Performance Simulation Association

[29] Zhou J, Vicario U, Tchounwou PB, Abdelghani AA, Ahmad F, Allen KJ & Arsanjani JJ (2004), Journal Browser. *International Journal of Environmental Research and Public Health* 1

[30] Hess-Kosa K (2016), *Indoor air quality: the latest sampling and analytical methods*. CRC Press

[31] Anon. (2000), *NIOSH pocket guide to chemical hazards*. DIANE Publishing

[32] Charles KE, Magee R, Won D & Lusztyk E (2005), Indoor air quality guidelines and standards. Institute for Research in Construction, National Research Council Canada

[33] Anon. (2000), Air quality guidelines for Europe.

[34] Anon. (2013), Standard 55-2013: thermal environmental conditions for human occupancy. ASHRAE

[35] Axley JW (2001), Application of natural ventilation for US commercial buildings–climate suitability, design strategies and methods, modeling studies. Gaithersburg, MD, NIST

[36] Kim G, Schaefer L, Lim TS & Kim JT (2013), Thermal comfort prediction of an underfloor air distribution system in a large indoor environment. *Energy and Buildings* 64, 323-331

[37] Nielsen PV, Allard F, Awbi HB, Davidson L & Schälin A (2007), Computational fluid dynamics in ventilation design. *International Journal of Ventilation* 6(3), 291-293

[38] Ho SH, Rosario L & Rahman MM (2009), Three-dimensional analysis for hospital operating room thermal comfort and contaminant removal. *Applied Thermal Engineering* 29(10), 2080-2092

[39] Setaih K, Hamza N, Mohammed M, Dudek S & Townshend T (2014), CFD modeling as a tool for assessing outdoor thermal comfort conditions in urban settings in hot arid climates. ITcon

[40] Villafruela J, Olmedo I, de Adana MR, Méndez C & Nielsen PV (2013), CFD analysis of the human exhalation flow using different boundary conditions and ventilation strategies. *Building and Environment* 62, 191-200

[41] Anon. (2011), *Fluent User's Manual.* ANSYS Inc.

[42] Franke J, Hellsten A, Schlunzen H & Carissimo B (2007), The Best Practise Guideline for the CFD simulation of flows in the urban environment. 5th International Symposium on Computational Wind Engineering