DEVELOPMENT AND VALIDATION OF THE CFD MODEL FOR NATURAL VENTILATION OF UNDERGROUND SHELTER

Azfarizal Mukhtar[1,*],Ng Khai Ching[1], Mohd. Zamri Yusoff[1]

[1] *Centre for Fluid Dynamics, College of Engineering, Universiti Tenaga Nasional (UNITEN), Putrajaya Campus, Jalan IKRAM-UNITEN, 43000 Kajang, Selangor*

[*]For correspondence; Tel. + (60) 193848425, E-mail: izalrajd@gmail.com

ABSTRACT: *Occupants of underground shelters often experience problems such as poor ventilation, high humidity, lack of natural lighting, as well as excessive heat. These problems may incur health issues on the occupants. This study aims to numerically determine the ventilation rate for a naturally-ventilated underground shelter. CFD is employed as the simulation tool in this study, however, careful verification and validation of the CFD results are required for the general acceptance of the model. The ventilation rate was studied extensively by using the finite volume method, as well as the Reynolds-Averaged Navier-Stokes (RANS) method. The realizable k-ε model was used to model flow turbulence and SIMPLEC algorithm was used for pressure - velocity coupling. The numerical results are compared with the available experimental data and good agreement was found. From this study, it is shown that CFD can be used to model the natural ventilation in the underground shelter and hence, it can be utilized as a guideline to solve some new issues related to underground shelter problems.*

Keywords: CFD Simulations, Underground Shelter, Validation, Natural Ventilation

1. INTRODUCTION

The need of ventilation systems has been recognised since a decade ago. In this regard, proper ventilation can ensure a proper circulation of fresh air inside the building, which would effectively remove carbon dioxide, heat and moistures. Thus, a good ventilation system is equally important for an underground shelter. As reported by Benardos et al. [1] and Flanigan & Gonzalez [2], occupant of underground shelters usually encounter some problems like poor ventilation system, poor access to natural lighting, high humidity, excessive heat and poor airflow within the structure.

According to the US Federal Emergency Management Agency [3], most underground shelters are equipped with a mechanical device to provide adequate ventilation rate instead of relying on the natural ventilation. Even so, there is a necessity for the application of natural ventilation if mechanical system fails [4], so that warm air could be removed and enough cooler outdoor air could be brought to maintain the occupant temperature-humidity conditions. Consequently, the failure of the ventilations systems will cause health issues among occupants [5], particularly symptoms of Sick Building Syndrome (SBS) [6].

As explained by Erdem [7], underground shelter could serve as a proper shelter and provide warmth to the occupants during different climates. Actually, the exploration of underground spaces for building construction has been existing since a long time ago such as the Yaodong cave houses in China since 206 BC, the Kandovan rock in Iran since the 13th Century, the Mesa Verde in Colorado since 7500 BC and the Coober Pedy in Australia since 1915 [1]. The latest discovery related to the underground space was an underground city in Derinkuyu, Turkey. It was found accidentally by a local native in 1963 while renovating his house. Researchers believe that it has been existing since 12,000 years ago. The facilities (e.g. ventilation shafts, residential quarters, schools, stores, cemeteries, and so forth) within the underground city can accommodate for at least 20,000 peoples. With a few historical pieces of evidence, it seems that underground spaces have a potential for human occupancy. However, the research is still on-going due to a few unresolved issues such as those mentioned above.

There are few published research papers which are related to underground spaces such as underground building (Shavadoon spaces) [8,9], ancient tomb [10,11], subway tunnel [12,13] and underground mines [14–16]. However, to date, research works related to natural ventilation in an underground shelter are somehow limited. Therefore, the main purpose of this paper is to determine the ventilation rate for the underground shelter via CFD. The numerical results were subsequently validated by the available experimental data and experimental data used are provided in the reference [17]. Consequently, CFD seems to be the perfect tool to determine the ventilation rate for the underground shelter [8,18]. In this regard, in this paper, the geometry used in the simulation (see Figure (1b)) was similar to the one used by King [17](see Figure (1a)). As a result, a CFD model of underground shelter was developed to predict the minimum ventilation rate. More details on these experimental parameters and measurements can also be found in King [17].

2. METHODOLOGY

As a reliable CFD model should be validated with the accurate data either from empirical or experimental [19], this paper presents a formal methodology (see Figure (2)) that supports the development and validation of a CFD model for natural ventilation for an underground shelter. In this light, Figure (2), explains the verification (orange colour) and validation processes (yellow colour) for the CFD model and how the parameter analysis (grey colour) can be performed to evaluate the influence of model boundary conditions on the results. Meanwhile, the green colour represents the outcomes and reliability of the CFD model. In other words, it can also represent the real environment.

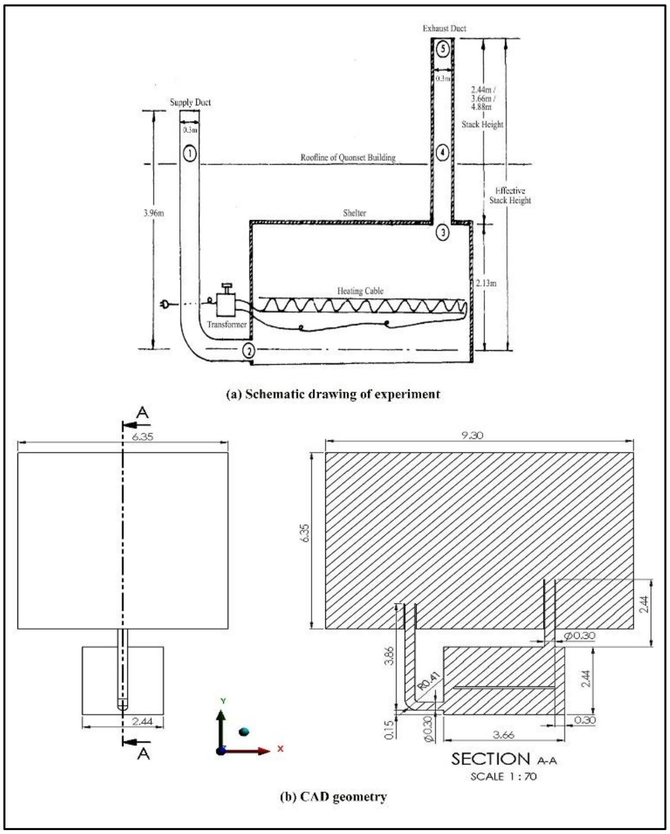

Figure (1) schematic drawing and geometry of underground shelter

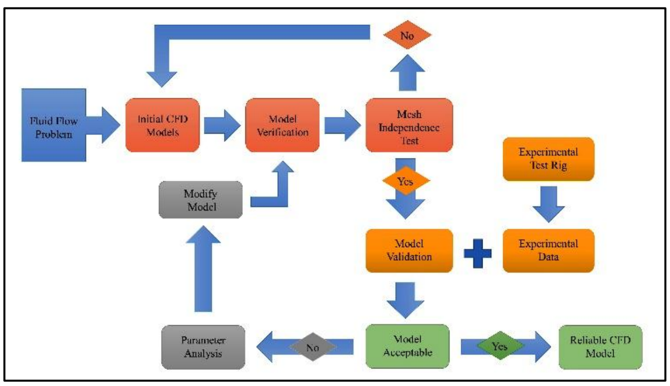

Figure (2) process of a reliable CFD model

3. CFD MODELS DEVELOPMENT

In the meantime, the use of CFD in solving the governing equations for fluid flow and thermal comfort in underground spaces has increased tremendously. However, these CFD results must be verified and validated before it can be used as an accurate instrument to perform product researches, development, and designs in the industry. In this regard, Chen & Srebric [20] and Oberkampf et al. [21] suggested some guidelines for the verification and validation of CFD analyses.

In this current study, commercial ANSYS Fluent R16.0 was employed as the CFD solver and the post-processor tool. It was decided to utilise the commercial software as the main tool for solving this complex flow problem, based on the fact that the main concern of the current work is to check the validity of the previously obtained experimental data with the more established commercial software.

As shown in Figure (1b), a rectangular region located at the top of the ventilation shafts (supply and exhaust duct) was designed to model the atmosphere in the experimental condition. This rectangular region was used to replace the method of directly applying an input boundary condition at the supply duct. The similar concept was adopted by Hazbei et al. [8] in his study where this concept allows the airflow to be induced through the supply duct to mimic the real experimental conditions. Consequently, there were few assumptions made in the current CFD study:

(1) The flow in the duct is steady and fully turbulent.
(2) No heat transfer (conduction and radiation) occurs at the ventilation shaft and underground shelter.
(3) Heat flux values are set according to the temperature difference (ΔT).
(4) Boussinesq approximation is used to model the buoyancy-driven flow.

3.1. Governing Equations

The general governing equations for the CFD used in this study are presented below:

(1) Conservation of mass

$$\frac{\partial \rho}{\partial t} + \nabla . \rho \vec{v} = 0 \qquad (1)$$

Here ρ is density, t is time and $\vec{v}$ is velocity vector.

(2) Conservation of momentum

$$\rho\left[\frac{\partial \vec{V}}{\partial t} + (\vec{V}\cdot\Delta)\vec{V}\right] = -\nabla p + \nabla\cdot\bar{\bar{\tau}} + \rho\vec{g} \qquad (2)$$

Here ρ is pressure, $\bar{\bar{\tau}}$ is the viscous stress tensor and $\vec{g}$ is gravitational acceleration. The terms $\rho\vec{g}$ at the RHS of Eq. (2) represents the buoyancy force.

(3) Conservation of energy

$$\rho\left[\frac{\partial E}{\partial t} + \nabla\cdot(E\vec{v})\right] + \frac{Dp}{Dt} = \nabla\cdot(k\nabla T) + \Phi + S_h \qquad (3)$$

where E is the total energy, t is the absolute temperature, k is the fluid thermal conductivity, S_h is a source term and Φ is the dissipation function representing the work done by the viscous forces.

3.2. Numerical tool, boundary conditions and computational mesh

Moreover, in this paper, the finite volume method was used to discretise the Reynolds-Averaged Navier-Stokes (RANS) equation and the equations were solved by using the steady pressured-based solver. The realizable k-ε model was applied to model the flow turbulence and the pressure - velocity coupling algorithm, SIMPLEC was used. Then, Pressure Equation is discretized using the second-order scheme while the other equations are discretized by using the second-order upwind schemes. Lastly, the solution is executed for approximately 300 iterations in order to attain the steady - state solution where the convergence criterion were 10^{-6} for the energy equation and 10^{-3} for other equations. The general features and parameter used for the simulation are presented in Table 1.

Table 1 general features and parameter used

Parameter	Unit	Value	Code Version	Fluent 16.0
Diameter of ventilation shaft	m	0.2	Solver	Pressured-based (steady-state)
Stack height	m	2.44	FVM	RANS
		3.66	Turbulence Modelling	Realizable k-ε
		4.88	Boussinesq parameters	28^0C
Outside Temperature	^{0}C	23	Pressure-Coupling	SIMPLEC
Air Velocity	m/s	2.24-3.58	Spatial discretization	Second-order Upwind
Heat Flux	W/m^2	70	Wall	No Slips

The boundary conditions shown in Fig. 3 are described below:

(1) *Domain Inlet*: The velocity inlet boundary condition was used. The velocity magnitude varied from 2.24m/s to 3.58 m/s and the temperature was fixed at 230C.

(2) *Domain Outlet*: Pressure outlet boundary conditions was used where the pressure value was prescribed as 0 Pa.

(3) *Wall*: For solution domain above ground, the walls were treated as free slip walls with a constant temperature of 23°C. On the other hand, the walls were treated as adiabatic, no-slip walls ($u_i = 0$). The wall roughness values were taken from Munson et al. [22]. Meanwhile, for the underground shelter, the walls were treated as adiabatic, no-slip walls ($u_i = 0$). The exterior surfaces of the heating cable were also treated as no-slip walls ($u_i = 0$) with the heat flux specified at 70 W/m^2.

Furthermore, the model was discretized in unstructured mesh (see Figure (4)), using tetrahedral elements which was developed using ICEM CFD R16.0. In this light, tetrahedral mesh is generally recommended when dealing with complex geometry. This is because the clustering of the cells in the flow domain is easier compared to the hexahedral mesh. In order to check the dependency of mesh size on the flow results, cases employing six different element sizes were simulated. The mesh-quality indices of the six different meshes are reported in Table 2

Table 2 mesh independence test

Case	Element size	Number of cells	Ventilation Rate (m^3/s)		%
			Experiment	CFD	
1	default	829731	0.0198	0.0115	41.92
2	0.5	843896		0.0176	11.11
3	0.2	1176160		0.0188	5.00
4	0.175	1348103		0.0193	2.53
5	0.15	1676790		0.0196	1.16
6	0.125	2435243		0.0196	0.96

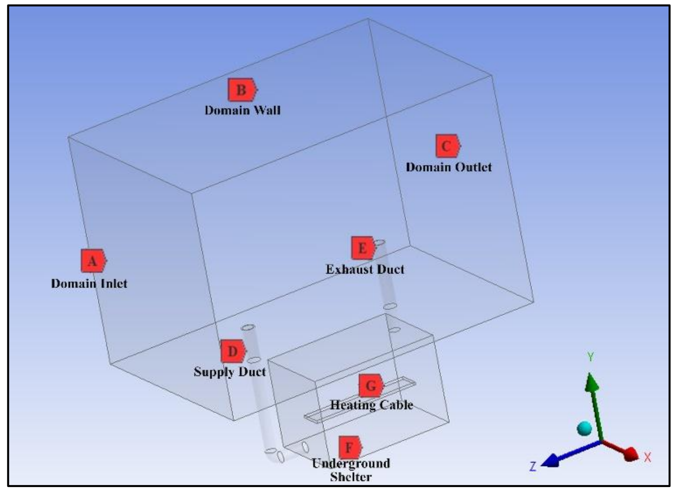

Figure (3) boundary conditions

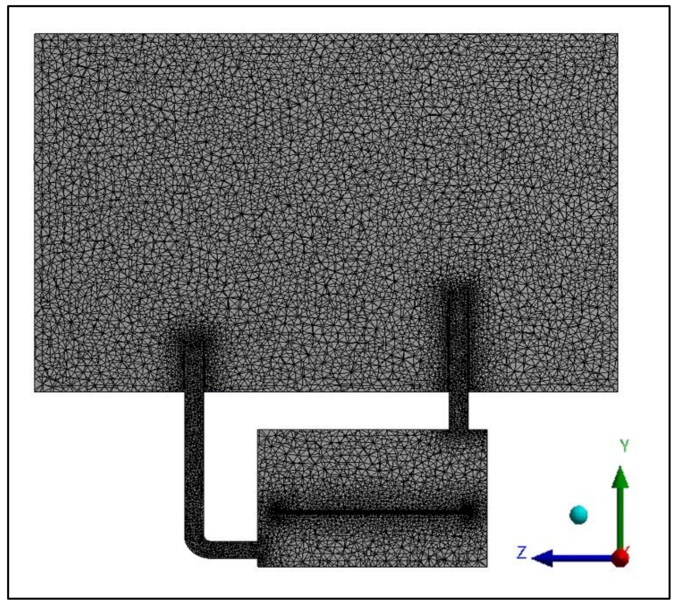

Figure (4) unstructured mesh

4. RESULT AND DISCUSSIONS

4.1. Mesh Independence Test and Validation

The mesh independence study was performed to verify the CFD model. It was carried out by utilising six different meshes where the mesh count up to 2.44 million cells were created. The numerical results of four cases reported in Table 2 are comparable with the experimental data, where the percentage relative errors ranged from 0.96% to 5%. Moreover, the result could be regarded as independent with the grid size. As illustrated in Figure (5) and Table 2, the mesh independence was achieved when the mesh employed 1.67 million cells. This is very close to the experimental result and it was further used for subsequent flow analysis of the validation process.

In the meantime, a systematic validation of the CFD model was also performed. Three different cases (various stack height) were generated and validated with the experimental data [17]. The result obtained are presented in Figure (6). According to the Figure (6), the relative errors ranged from 1.01% to 7.72%. This means that the current CFD models exhibit the error level of within 10%. It can be justified that the simulated result agreed well with the experimental data. However, apart from the numerical errors derived by the CFD model, it is equally important to note that certain measurement errors might be associated with the experimental data.

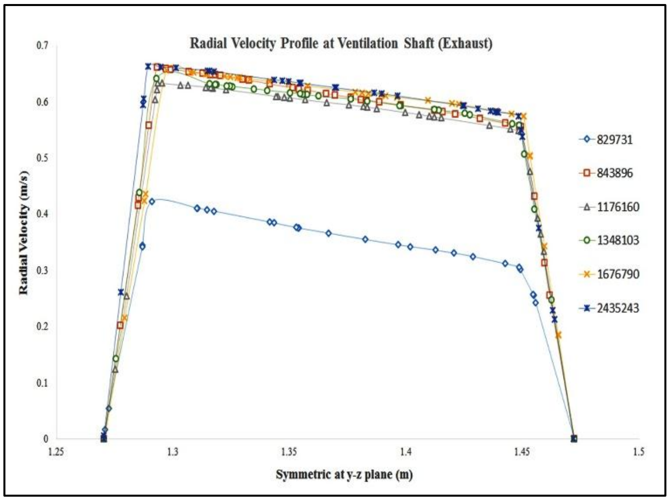

Figure (5) CFD verification

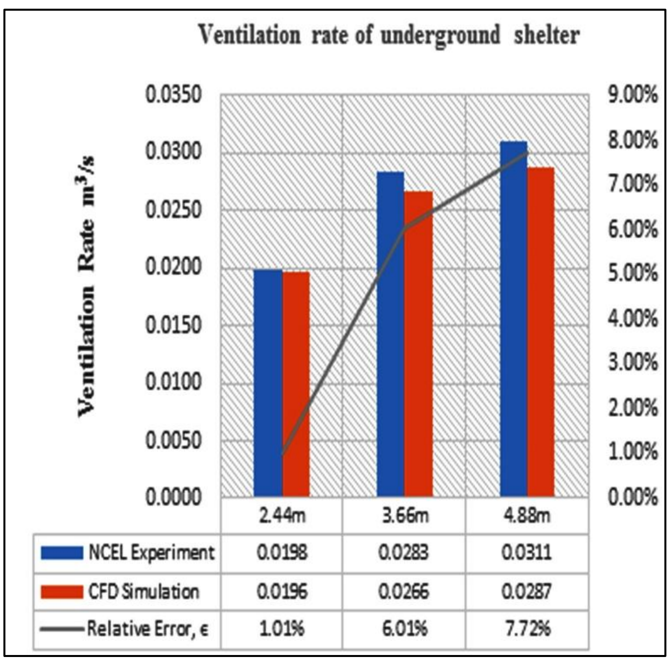

Figure (6) CFD validation

4.2. Volume flow rate (ventilation rate)

From the current study (see Figure (6)), the largest ventilation rate ($0.0287m^3/s$) was obtained by exhaust duct, 4.88m. While, the lowest ventilation rate ($0.0196m^3/s$) was obtained by exhaust duct, 2.44m. Interestingly, this is also observed experimentally by King [17]. Note also that higher stack height (exhaust duct) significantly increase the ventilation rate.

5. CONCLUSION

This study has validated the CFD methodology to model the natural ventilation of an underground shelter, as previously experimentally investigated in the US Navy Civil Engineering laboratory (NCEL). Consequently, the simulation results, has shown a good agreement with the experimental data. Most importantly, the numerical settings derived from the current work could be served as a useful guideline for future studies in natural ventilation. Based on the current results, it appears that CFD can become one of the reliable tools for analysing natural ventilation problems.

6. ACKNOWLEDGMENT

This study was supported by a scholarship provided by Yayasan Tenaga Nasional (YTN) to the first author.

7. REFERENCES

[1] A. Benardos, I. Athanasiadis, N. Katsoulakos, Modern earth sheltered constructions: A paradigm of green engineering, *Tunn. Undergr. Sp. Technol*. **41**:46–52(2014).

[2] F.M. Flanigan, J. Juan O.Gonzalez, Moisture in Survival Shelters, Florida Univ Gainesv. Eng. Ind. Exp. Stn. 0–11(1967).

[3] FEMA453, Risk Management Series Design Guidance for Shelters and Safe Rooms, (2006).

[4] T.L. Holthusen, ed., The Potential of Earth-sheltered and Underground Space: Today's Resource for Tomorrow's Space and Energy Viability, (2013).

[5] C.H. Kearny, L. Mims, Nuclear war survival skills, Oregon Institute of Science and Medicine, (1987).

[6] J. Sundell, H. Levin, W.W. Nazaroff, W.S. Cain, W.J. Fisk, D.T. Grimsrud, F. Gyntelberg, Y. Li, A.K. Persily, A.C. Pickering, J.M. Samet, J.D. Spengler, S.T. Taylor, C.J. Weschler, Ventilation rates and health: Multidisciplinary review of the scientific literature, *Indoor Air*. **21**:191–204(2011).

[7] A. Erdem, Subterranean space use in Cappadocia: The Uchisar example, *Tunn. Undergr. Sp. Technol*. **23**:492–499(2008).

[8] M. Hazbei, O. Nematollahi, M. Behnia, Z. Adib, Reduction of energy consumption using passive architecture in hot and humid climates, *Tunn. Undergr. Sp. Technol*. **47**:16–27(2015).

[9] H. Moradi, H. Eskandari, An experimental and numerical investigation of Shovadan heating and cooling operation, *Renew. Energy*. **48**:364–368. (2012)

[10] D. Gribble, Natural ventilation , harnessed by New Kingdom Egyptian tomb builders , may explain the changed floor levels in the Valley of the Kings tomb KV5, *Tunn. Undergr. Sp. Technol. Inc. Trenchless Technol. Res*. **24**:62–65(2009).

[11] O.A.A.A. Abdel-Aziz, E.E. Khalil, Air flow regimes and thermal patterns in climatized tombs in the valley of the kings, *43rd AIAA Aerosp. Sci. Meet. Exhib*. AIAA-2005-1209–1216(2005).

[12] F.-D. Yuan, S.-J. You, CFD simulation and optimization of the ventilation for subway side-platform, *Tunn. Undergr. Sp. Technol.* **22**:474–482(2007).

[13] Y.D. Huang, X.L. Gong, Y.J. Peng, X.Y. Lin, C.N. Kim, Effects of the ventilation duct arrangement and duct geometry on ventilation performance in a subway tunnel, *Tunn. Undergr. Sp. Technol.* **26**:725–733(2011).

[14] A.P. Sasmito, E. Birgersson, H.C. Ly, A.S. Mujumdar, Some approaches to improve ventilation system in underground coal mines environment - A computational fluid dynamic study, *Tunn. Undergr. Sp. Technol*. **34**:82–95(2013).

[15] G. Xu, E.C. Jong, K.D. Luxbacher, S.A. Ragab, M.E. Karmis, Remote characterization of ventilation systems using tracer gas and CFD in an underground mine, *Saf. Sci.* **74**:140–149(2015).

[16] T. Ren, Z. Wang, G. Cooper, CFD modelling of ventilation and dust flow behaviour above an underground bin and the design of an innovative dust mitigation system, *Tunn. Undergr. Sp. Technol.* **41**:241–254(2014).

[17] J.C.King, Gravity Ventilation of Underground Shelters, Nav. Civ. Eng. Lab Port Hueneme CA. **1**:26(1965).

[18] P.E.E. Khalil, 21st century CFD prediction of flow regimes and thermal patterns in 15th century BC tombs of the Valley of Kings, *Am. Inst. Aeronaut. Astronaut*. **129**:1–8(2006).

[19] M. Hajdukiewicz, M. Geron, M.M. Keane, Formal calibration methodology for CFD models of naturally ventilated indoor environments, *Build. Environ*. 59:290–302(2013).

[20] Q. Chen, J. Srebric, A procedure for verification, validation, and reporting of indoor environment CFD analyses, *HVAC&R Res. ASHRAE Trans*. **8**:201–216(2002).

[21] W.L. Oberkampf, M. Sindir, A. Conlisk, Guide for the verification and validation of computational fluid dynamics simulations, *Am Inst. Aeronaut. Astronaut*. G-077-1998-1–29(1998).

[22] B.R. Munson, T.H. Okiishi, W.W. Huebsch, A.P. Rothmayer, Fundamentals of fluid mechanics, 7th Editio, John Wiley & Sons, Inc., (2013).

* For correspondence; Tel. + (60) 193848425, E-mail: izalrajd@gmail.com