

Simulation of naturally ventilated underground car park with CFD

Truong Tich Thien*, Nguyen Thi Huong Mai



Use your smartphone to scan this QR code and download this article

ABSTRACT

The basement car park is a popular solution to effectively use urban land, especially in commercial centers and apartments. However, the situation of apartment fire and explosion is a hot problem, partly due to the tunnel ventilation has not met the requirements of fire safety. Therefore, the design of the car park basement ensures fire safety as well as bring comfort to people. In this study, the computational fluid dynamics (CFD) method has been applied to determine the pressure and velocity intensity for buildings that detect residuals in architecture, thereby improving and providing an optimal solution. The Ansys CFX commercial software is used to simulate the ventilation process in a parking basement. In Ansys CFX, the Shear-Stress Transport turbulence model is used to determine the temperature and velocity of the tunnel and calculate the level of physical comfort according to ADPI standard. Firstly, a simulation problem of ventilation in the workshop room is also performed to verify the reliability. Finally, the problem of basement temperature and wind velocity by natural ventilation method will be analyzed and evaluated in accordance with ADPI standard and carbon monoxide concentration with WHO standard to identify areas of unsatisfactory temperature and velocity to reasonably adjust and propose other suitable ventilation options. The results from this study present a promising natural ventilation analysis procedure and technique for car parking basements of tall buildings in major cities.

Key words: natural, ventilation, basement, computational, fluid, dynamics, Ansys, air, diffusion, performance, index

INTRODUCTION

Nowadays, the speed of urbanization is increasing rapidly, so the urban land area is fully utilized to build high-rise buildings, apartments, and commercial centers and thus, the parking basement also become more popular. However, apartment fire and explosion, especially car fire and explosion is an extremely important issue that must be concerned in construction design. Therefore, it is essential to design an effective ventilation system in the parking basement when a fire occurs. When building up the car park, the importance is not only a reasonable architecture but also the ventilation and air quality of the tunnel because it directly affects human health.

Decades ago, scientists had studied the solution to ventilate the car park. The computational fluid dynamics (CFD) method is also applied to determine the pressure and velocity intensity for buildings that detect residuals in architecture, thereby improving and providing a superior solution. Many studies related to this issue have been published internationally. Jiang¹ also investigated natural ventilation by using Reynolds Averaged Navier - Stokes turbulence model (RANS). Khalil² also examined the distribution of CO emissions from buses in a basement in Cairo using ANSYS FLUENT software.

In this paper, Ansys CFX commercial software is used to simulate the ventilation process in a parking basement. In addition, a simulation problem of ventilation in the office room is also performed to verify the reliability. In Ansys CFX, the Shear-Stress Transport turbulence model is used to determine the temperature and velocity of the tunnel and calculate the level of physical comfort according to ADPI standard.

MATERIALS AND METHODS

Ventilation Theory

Natural Ventilation

Natural ventilation occurs when the air in a space is exchanged outside without using mechanical systems such as fans. Common types of natural ventilation are made through open windows but can also be achieved through differences in temperature and pressure between spaces. However, opening a window or grounded structure is not a good choice for basement ventilation or underground structures.

Mechanical Ventilation

Mechanical ventilation or forced system is through an air handling unit or directly into space by fans. Local

Department of Engineering Mechanics,
Faculty of Applied Science, Ho Chi Minh
city University of Technology,
VNU-HCM, Vietnam

Correspondence

Truong Tich Thien, Department of
Engineering Mechanics, Faculty of
Applied Science, Ho Chi Minh city
University of Technology, VNU-HCM,
Vietnam

Email: ttruong@hcmut.edu.vn

History

- Received: 20-3-2020
- Accepted: 11-11-2020
- Published: 22-11-2020

DOI : 10.32508/stdjet.v3iS13.686



Copyright

© VNU-HCM Press. This is an open-access article distributed under the terms of the Creative Commons Attribution 4.0 International license.



Cite this article : Thien T T, Mai N T H. **Simulation of naturally ventilated underground car park with CFD.** *Sci. Tech. Dev. J. – Engineering and Technology*; 3(S13):SI22-SI28.

exhaust fans can enhance warmth or natural ventilation, thereby increasing air flow.

Analytical Methods

Commercial software ANSYS CFX is the key tool for calculating velocity, temperature and then comparing it to the APDI standard³. The EDT (Effective Draft Temperature) indicator is calculated by the following formula:

$$EDT(\theta) = T_i - T_a - 8,0(V_i - 0,15)^\circ C \quad (1)$$

Where:

T_i : The local temperature of air flow at examinational points;

T_a : The average temperature of examinational space;

V_i : The local velocity of air flow;

The APDI is then interpreted as the ratio between the total number of points with the EDT index within the standard range [-1.7°C, 1.1°C] and the airflow velocity below or equal to 0.35 m/s with the total number of points to be measured. The human body only feels comfortable in spaces with an ADPI index of 0.8 (80%) or more.

Apply the Navier - Stokes equation (Polyanin, Kutepov, Vyazmin, & Kazenin, 2001) to simulate the flow of fluid. Equations are built from the conservation of the mass of momentum and energy written for any volume under consideration. According to Polyanin and *et al.*⁴, the most general form of the system of equations is:

$$\rho \left(\frac{\partial v}{\partial t} + v \nabla v \right) = -\nabla p + \nabla \cdot T + f \quad (2)$$

Where:

The right side of the equation is the sum of the forces acting on the object with:

p: pressure;

$\nabla \cdot T$: represents the deformation forces in fluid, usually due to the effect of viscosity;

f: volume force;

The left side of the equation:

ρ : density;

v: velocity vector;

$\partial v / \partial t$: instant acceleration;

$v \cdot \nabla v$: convection acceleration;

This is the law of conservation of momentum for a fluid, applying Newton's second law to the environment continuously. According to Polyanin and *et al.*⁴, this equation is written as a material derivative:

$$\rho \frac{Dv}{Dt} = -\nabla p + \nabla \cdot T + f \quad (3)$$

The Problem Model

Room Study Problem

The problem is based on the Workshop Room Study of ANSYS CFX⁵ as shown in Figure 1. The velocity and temperature field between the two models after re-modeling the sample problem on ANSYS CFX 19.1 is shown in Figure 2.

The Car Park Basement Problem

The model of parking basement based on the car park basement size of the Kientrucmoi site with a size of 14.6 x 26.6 m. The capacity of the car park holds 28 motorbikes of 2,285 x 0.9 x 1.41 m and each vehicle has an exhaust pipe with a diameter of 56 mm arranged at the rear of the vehicle modeled with SOLIDWORKS and the position of the boundary areas is shown in Figure 2. The problem applies the Shear Stress Transport (SST) model, which is in a stable state and is meshed with a particle size of 0.5 m and divided by the approximation method. The model is divided into 3,625,438 nodes and 2,467,591 elements. The distribution of temperature and velocity of the car park in the case of 16 vehicles are obtained after simulation process. The natural wind speed of 0.15 m/s. The temperature of the basement is 290C and the temperature of the air inlet is 270C. The mass flow of each exhaust pipe is 17bhp air temperature is 30 °C, air pressure does not use industrial fans.

RESULTS AND DISCUSSIONS

Room Study Model

Take the value of velocity and temperature at certain points from the results of the two problems to compare and evaluate the error.

The degree of error of temperature and velocity of the two problems is shown in Figure 3.

The difference temperature and velocity results between two problems is calculated according to the following formula:

$$E = \frac{\sum_{i=1}^n |X_{solution}^i - X_{model}^i|}{\sum_{i=1}^n X_{model}^i} \times 100\% \quad (4)$$

Where:

X: The considered object

It is easy to see that the temperature at 20 specific points between the two problems has no big difference, with the average error value of 0.080965%. While the speed at these points is quite different, the average error value is 8.724897%. This difference comes mainly from the version of Ansys software, due to the differences between the Reynolds coefficient and the other coefficients between the two versions 12.0 and 19.1.

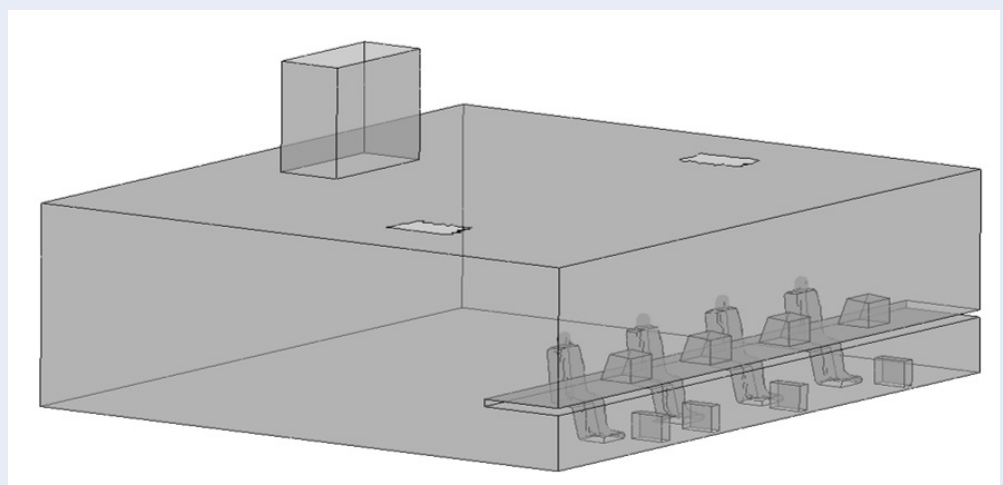


Figure 1: Room study 3D model

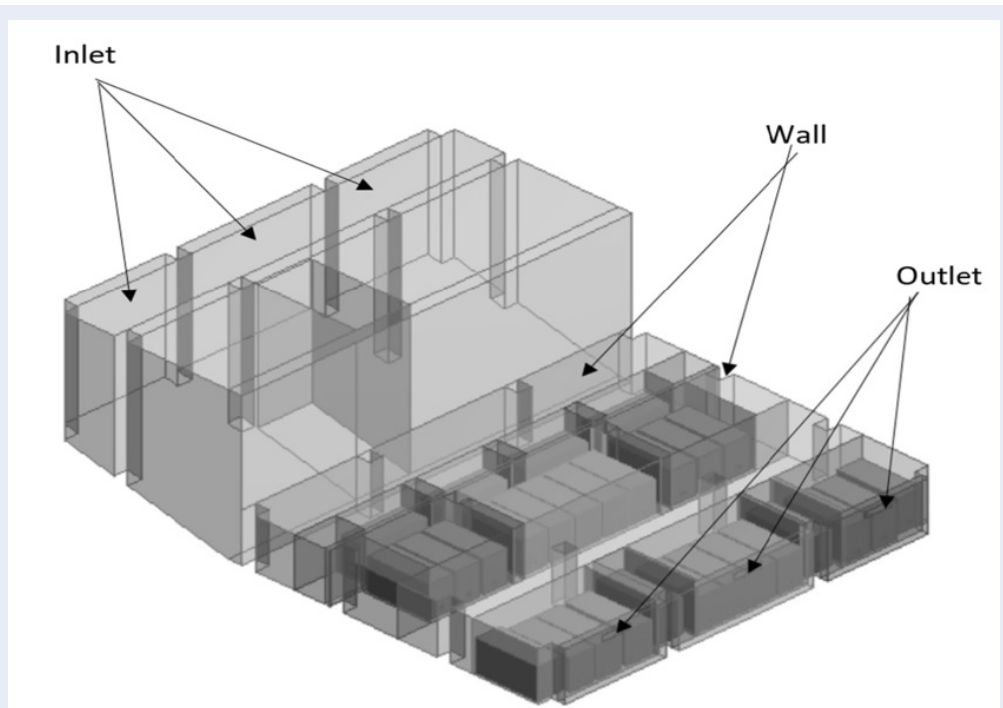


Figure 2: Car park basement 3D model

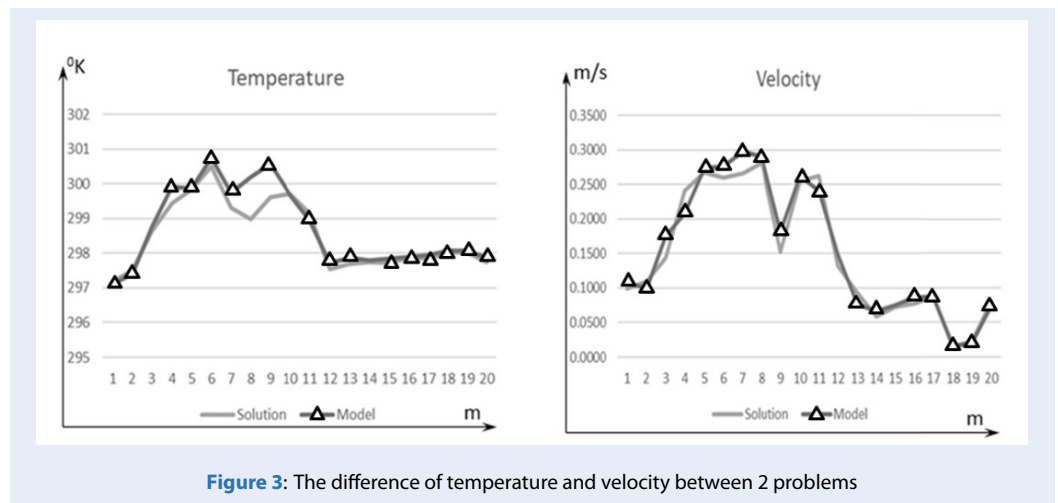


Figure 3: The difference of temperature and velocity between 2 problems

Car Park Basement Model

The temperature and velocity fields and CO volume fraction are shown in the Figures 4, 5 and 6. Through simulation results from Figure 4 and Figure 6, it can be seen that the high temperature zone is also the area with high concentration of CO.

Take 20 points in the entire air flow to calculate and compare the APDI standard, the velocity, temperature and EDT index of the points shown in Table 1. The average temperature of the air flow in the basement is 30.151 °C.

The result is that 17/20 points reach the EDT index within [-1.7°C, 1.1°C], the remaining point 16,19 and 20 are not yet achieved, equivalent to 85% of the APDI standard. Therefore humans can be comfortable in this air condition.

Besides, the max CO concentration of this space is just 6.375e-6 which showed at Figure 6, so it is good for human health according to WHO standards⁶.

CONCLUSIONS

The present study has shown the safety and rationality in the design of the car park to ensure both mechanical and biochemical safety by considering factors related to APDI and WHO standards. Besides, the topic still has a number of points left unresolved, namely, only research on carbon monoxide concentration, not considering other harmful emissions and not yet diversified in vehicle types. The simulation results visually show the temperature and CO distribution in the parking basement. From there, better designs for ventilation can be made based on current standards.

ACKNOWLEDGEMENTS

The research was funded by University of Technology, Vietnam National University – Ho Chi Minh City

in the framework of the project code SVCQ - 2018 - KHUD - 114. The authors sincerely thank University of Technology, Vietnam National University – Ho Chi Minh City for creating everything favorable conditions to carry out this research.

LIST OF ABBREVIATIONS

CFD: Computational Fluid Dynamics;
 RANS: Reynolds Averaged Navier - Stokes;
 CFX: A Computational Fluid Dynamics Software Program Solutions;
 ADPI: Air Diffusion Performance Index;
 EDT: Effective Draft Temperature.

CONFLICT OF INTEREST

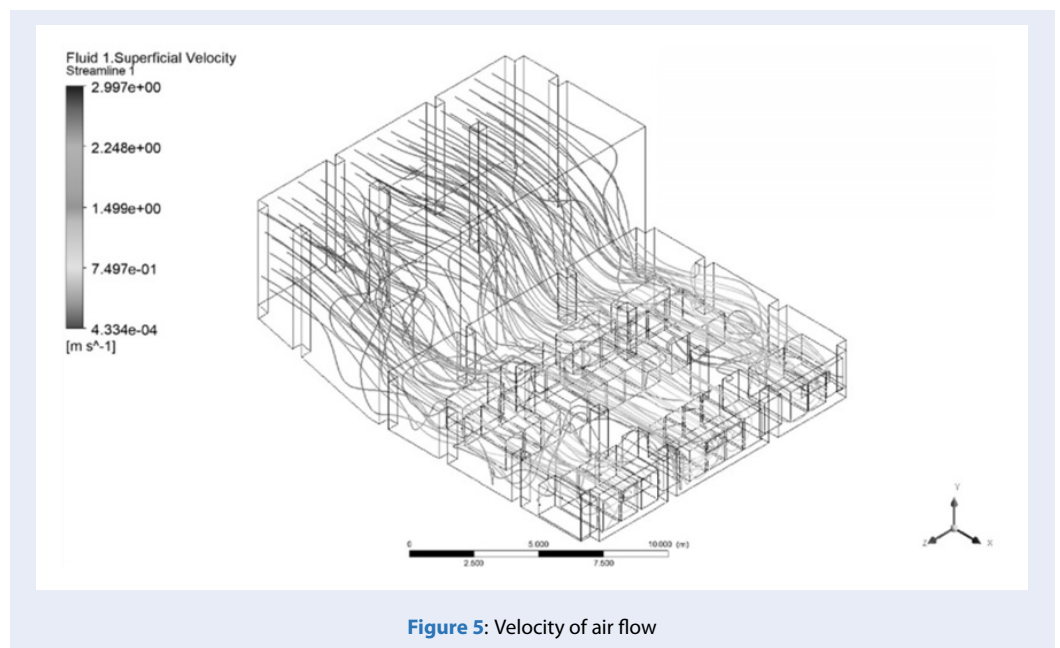
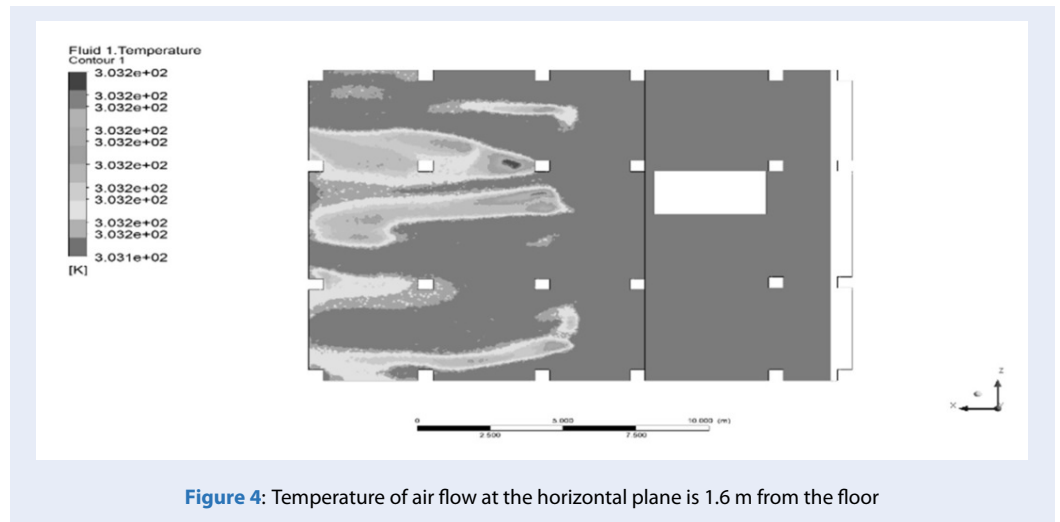
We declare that there is no conflict of whatsoever involved in publishing this research.

AUTHOR CONTRIBUTIONS

All authors contributed equally to this work. All authors have read and agreed to the published version of the manuscript.

REFERENCES

- Jiang Y, Allocca C, Chen Q. Validation of CFD simulations for natural ventilation. *Int J of Ventilation*. 2004;4(2):359–369. Available from: <https://doi.org/10.1080/14733315.2004.11683678>.
- E, Y, et al. Investigation of ventilation system performance of tahreer car park using CFD. *J of Energy*. 2015;2(2):81–89.
- Liu A. Air Diffusion Performance Index (ADPI) of diffusers for heating mode. *Building and Environment*. 2015;8:215–223. Available from: <https://doi.org/10.1016/j.buildenv.2015.01.021>.
- Polyanin A, Kutepov A, Vyazmin A, Kazenin D. *Hydrodynamics, mass and heat transfer in chemical engineering*. 2001; Available from: <https://doi.org/10.1201/9781420024517>.
- ANSYS. *Workshop 3: Room Temperature Study, Introduction to ANSYS CFX*. 2010;.
- WHO. *WHO Guidelines for indoor air quality*. 2010;.



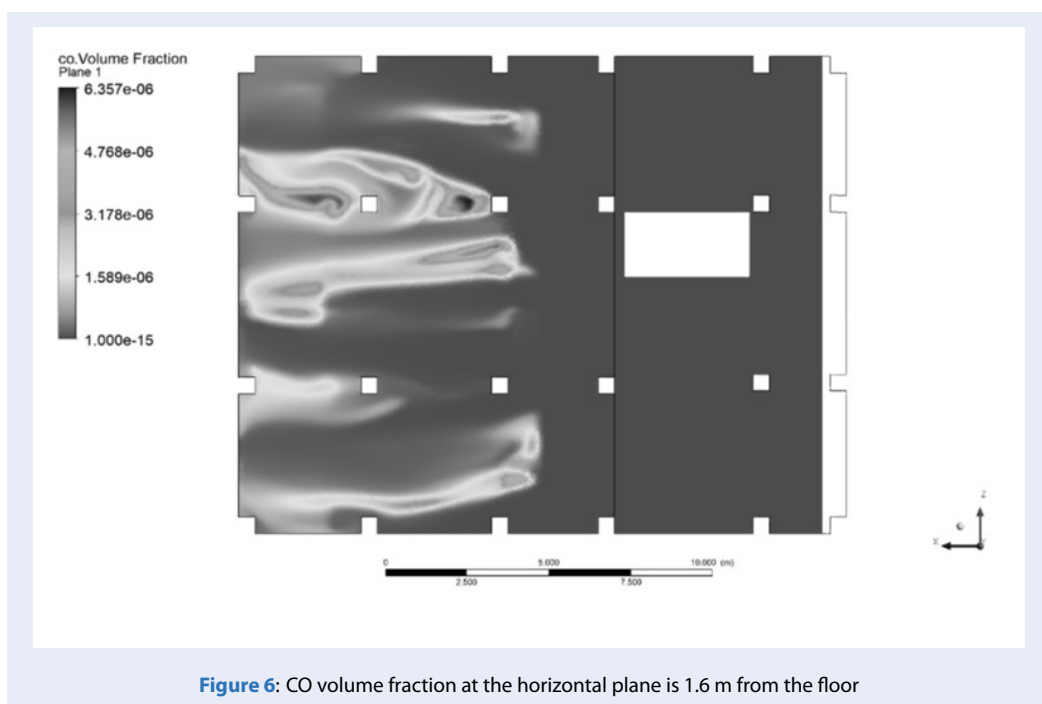


Table 1: Parameters of velocity, temperature and EDT

Number	Coordinates	Velocity, [m/s]	Temperature, [°C]	EDT
1	(15,1.6,-3)	0.303197	30.156	-1.220576
2	(15,1.6,-8)	0.26847	30.157	-0.94176
3	(15,1.6,-11)	0.294219	30.162	-1.142752
4	(15,1.6,-4)	0.183346	30.156	-0.261768
5	(15,1.6,-5)	0.227297	30.158	-0.611376
6	(13.5;1.6,-4)	0.164089	30.172	-0.091712
7	(13.5,1.6,-5)	0.248349	30.162	-0.775792
8	(13.5,1.6,-8)	0.299956	30.168	-1.182648
9	(13.5,1.6,-10)	0.137214	30.158	0.109288
10	(14,1.6,-4)	0.152628	29.155	-1.017024
11	(14,1.6,-5)	0.244363	30.16	-0.745904
12	(14,1.6,-8)	0.28114	30.165	-1.03512
13	(14,1.6,-10)	0.100534	30.157	0.401728
14	(14.5,1.6,-4)	0.209452	30.169	-0.457616
15	(14.5,1.6,-5)	0.262314	30.155	-0.894512
16	(14.5,1.6,-8)	0.276204	27.02	-4.140632
17	(14.5,1.6,-11)	0.330971	30.15	-1.448768
18	(8,1.6,-5)	0.1228	29.133	-0.8004
19	(8,1.6,-6)	0.257783	28.99	-2.023264
20	(8,1.6,-7)	0.295832	27.56	-3.757656