Penerbit
Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 8, No. 1. Pages 11-19, 2015

Large Eddy Simulations of Pollutant Dispersion in a Street Canyon

M. N. A. Mohd Yazid

Department of Thermo-Fluids, Faculty of Mechanical, Engineering, UniversitiTechnologi Malaysia, 8310 Skudai, Johor Bahru, Malaysia
afiqwitri@gmail.com

Abstract – *Flow structure and pollutant dispersion in a 3D symmetric street canyon are numerically investigated using a computational fluid dynamics (CFD) model. The analyzed parameter is Reynolds number. The studied canyons is regular canyon with aspect ratio equal to one. The Reynolds number used are 9000, 14200, 19200 and 30700 where the free stream velocity are 0.68m/s, 1.07m/s, 1.45m/s and 2.32 m/s. Transport equation for CO and air is included and is assumed as non-reactive. The source of pollutant emission is assumed only from vehicular fumes situated at ground level. The best meshing size used in this simulation is 547050 which is used to run this simulation using large eddy simulation (LES). Vortex's pattern and strength are dependent of velocity at the roof level. Analysis on the pollutant concentration shows that high Reynolds number will give the best air ventilation for street canyon subsequently decreased the pollutant concentration level.* ***Copyright © 2014 Penerbit Akademia Baru - All rights reserved.***

Keywords: Street Canyon, Turbulent Flow, Pollutant Dispersion, Large Eddy Simulation

1.0 INTRODUCTION

Investigation on pollution dispersion in urban street canyon has been investigated for decades. Different variables such as street geometry, wind conditions, and city climate will produced distinct environment inside the canyon itself, and these variables has been the centre of attention for the researchers as described by Yazid et al. [1]. Nicholson [2] describes a street canyon is a relatively narrow street between two buildings which are arranged in continuous. Street canyon that is surrounded by tall buildings reduces the ventilation of air inside and when combined with heavy traffic that emits pollutants, it creates poor air quality, which is a major concern in urban area.

In the past two decades, researches have been investigating the air flow and pollutant dispersion in street canyon by several methods such as field observation, small-scale experiments (e.g. wind tunnels) and computational fluid dynamics (CFD). Amongst the important aspect in understanding the flow structure and pollutant dispersion in an urban street canyon are the bulk effect from buildings on mean and fluctuating properties of air flow and pollutant concentration. The largest sources of urban air pollution in town and cities throughout the world are vehicular emissions as the road traffic continues to grow annually, which further complicates urban air pollution.

Carbon monoxide (CO) is one of the hazardous gases, which is emitted by ground vehicles. From modelling point of view, CO is preferred as the pollutant species for the simulation of pollutant dispersion in a street canyon compared with other vehicle exhaust pollutants due to the fact that the chemical reaction time within the street canyon is quite long, which is within a few weeks. Therefore, CO can be assume as non-reactive and thus reduces the mathematical formulation involved in CFD. Smagorinsky [3] introduced large eddy simulation (LES) as a mathematical model for turbulence used in CFD. Walton and Cheng [4] described the LES is able to provide detailed information on the turbulence statistics of the flow and offer the potential for improved accuracy and in the physical process of pollution dispersion, it allows the greater insight. On other hand, LES take a long time to achieve the results if compared with the Reynolds-averaged Navier-Stokes (RANS) but the result is more accurate.

In this study, the effect of Reynolds number on wind flow and pollutant dispersion in regular canyon with aspect ratio equal to one are investigated using commercial software ANSYS FLUENT v14. The Reynolds number used are 9000, 14200, 19200 and 30700 where the free stream velocity are 0.68 m/s, 1.07 m/s, 1.45 m/s and 2.32 m/s. The results from this study provide valuable information with regards to the effect of wind speed on airflow and pollutant dispersion in a street canyon.

2.0 METHODOLOGY

Filtered Continuity and Momentum Equation. The continuity and momentum equations in LES are shown in Equation (1) and (2) respectively. The overbar indicates spatial filtering, and not time-averaging as is the case of RANS. It is worth identifying that the filtered momentum equation is similar to the RANS equation. The spatial filtering is integration just like time-averaging, the difference being that the integration is in space and not over time as in the case of RANS.

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial t}\left(\rho \bar{u}_i\right)=0 \tag{1}$$

$$\frac{\partial}{\partial t}+\left(\rho \bar{u}_i\right)+\frac{\partial}{\partial x_j}\left(\rho \bar{u}_i \bar{u}_j\right)=\frac{\partial}{\partial x_j}\left(\sigma_{ij}\right)-\frac{\partial \bar{p}}{\partial x_i}-\frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

Species Transport Equation. FLUENT predicts the local mass fraction of each species, through the solution of a convection-diffusion equation for i^{th} species. This conservation equation takes the following general form:

$$\frac{d}{dt}\left(\rho Y_i\right)+\nabla.\left(\rho \bar{v} Y_i=-\nabla.\vec{J_i}+R_i+S_i\right) \tag{3}$$

where in turbulent flow,

$$\vec{J_i}=-\left(\rho D_{i,m}+\frac{\mu_t}{Sc_t}\right)\nabla Y_i \tag{4}$$

where R_i is the net rate of production of species i by chemical reaction. S_i is the rate of creation by addition from the dispersed phase. Sc_t is the turbulent Schmidt number.

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 8, No. 1. Pages 11-19, 2015

3.0 MODEL CONFIGURATION

In this study, three dimensional of street canyon are considered with the street aspect ratio is equal to one, where the height and width of the cavity are equal to 0.2 m as shown in Figure 1.Besides that, the street canyon model is assumed to be under isothermal atmospheric conditions which means the temperature at each wall like leeward, windward, and ground have the same temperature which is equal to 23 ºC. The incoming wind flow profile is equal to the wind tunnel study by Allegrini et al. [5] with the speeds ranges from 0.68 to 2.32 m/s. A pollutant source in volume is placed at the middle of street canyon having a cross sectional area equal to 0.004 $m^2$and projected along the street length was considered in this study to account for pollutant species. The working fluid is air, which is assumed to be incompressible at 23 ºC having a density, ρ = 1.1925 kg/m^3 and dynamic viscosity, $\mu = 1.835 \times 10^{-5}$ kg/ms. A volume source is demarcated within the street canyon along the middle of street canyon ground, ejected at a constant volumetric rate representing a continuous pollutant emitted from a busy traffic.

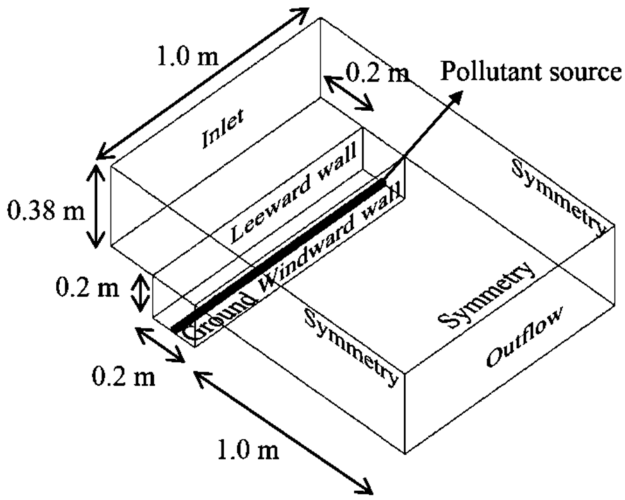


Figure 1: Geometry modelling, dimension of model and surface labels

4.0 RESULT AND DISCUSSION

Validation. Before proceeding to do simulation for different Reynolds numbers and constant aspect ratio which is equal to one, validation was initially conducted to ensure that the prediction using the numerical model was accurate. The validation is validated with wind tunnel measurement done by Allegrini et al. [5].

Mesh Independence Study. Mesh independence study is done to ensure that the result is independent of the mesh. In practice, the number of mesh is increase until there are no differences in the result obtained. This step was done for each Reynolds number with street aspect ratio equal to one, which is constant in this study. In this study, several numbers of meshes were chosen, which are 20 cells, 40 cells and 60 cells across the horizontal and vertical of street canyon width and height respectively. The parameter that used in mesh

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 8, No. 1. Pages 11-19, 2015

independence study is velocity because wind tunnel measurement done by Allegrini et al. [5] provides the results for velocity profiles on vertical centreline of street canyon.

Vertical line is drawn inside the street canyon after the simulation has achieved convergence. Graph of position in *y*-axis against *x*-velocity (streamwise velocity) is plotted. This step is repeated for different meshes which are 20 cells, 40 cells, and 60 cells. Each cell has the different total number of cells. As the number of cells increase, the total number of cells also will be increasing in each cell. From Figure2, it shown that velocity is considered had achieved mesh independent at mesh size of 547050 or 40 cells per cavity's length since there is little difference in the *x*-velocity along the vertical line from one mesh to another.

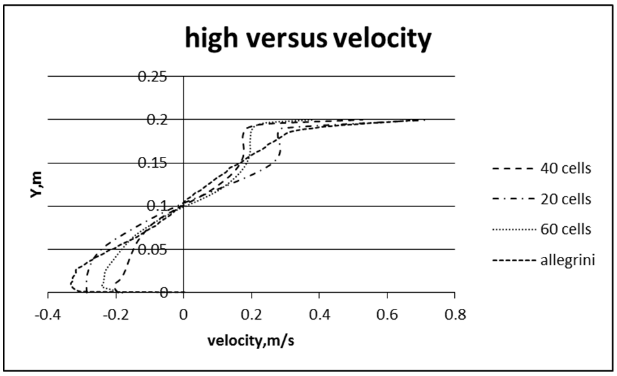


Figure 2: High against Velocity for different Cells.

Flow Structure. Canyon flow analysis has been originally discussed by Oke [6]and flow across canyons of different aspect ratio (*H*/*W*) is categorized into three regimes namely isolated roughness flow (IRF), wake interference flow (WIF) and skimming flow (SF). In numerical analysis canyon airflows are commonly discussed (Huang *et al*. [7]; Sini *et al*. [8]) and many research findings have attained the three regimes as mentioned by Oke [6].The flow structure in the cavity is affected by the geometry of the canyon and velocity of air flowing at roof level.

Table 1 show contour of velocity streamline within a street canyon at different Reynolds numbers. Generally a vortex which is a spiral motion of fluid within limited area, will appeared in street canyon as a result of wind flow at the roof level. The number and shape of vortex depends on the wind velocity as well as the containment area. Based on the velocity contour, a vortex is formed in the middle of street canyon at Re = 9000. Besides, a secondary vortex formed at the bottom of backwind building. The velocity of the flow depends on the distance from the walls or ground to the centroid of vortex that is usually formed at the middle of street canyon. As the distance away from the vortex centroid increases, the velocity also increase. As for the secondary vortex, the air flow rotates in a counter clockwise direction whilst for the primary vortex, the airflow rotates in a clockwise direction.

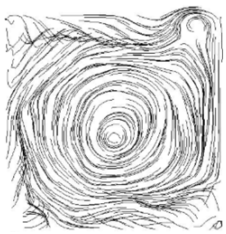
Besides that, different air speed also affects the vortex intensity. When the velocity in the street canyon increases, the intensification of vortex is subsequently increase. At Re=14200, the formation of the vortex is similar to the case at lower Reynolds number (Re = 9000),

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 8, No. 1. Pages 11-19, 2015

except that the vortex centroid is slightly shifted to the right of the street canyon. The reason to the dislocation of vortex centroid is due to the increase in velocity. Usually, the maximum velocity within the street canyon is always at the roof top. In this study however, at Re=14200, the highest velocity was found to be at the right top corner of windward wall (not shown) because the wind approached the windward wall first before it enters the canyon at a speed of 1.07 m/s.

The different in freestream velocity for each Reynolds number did not significantly changes the flow structure except for the velocity. The wind speed within the street canyon is increase at higher Reynolds number. By comparing the current results with wind tunnel experiment done by Allegrini et al. [5], the result of this study is in agreement with the wind tunnel study.

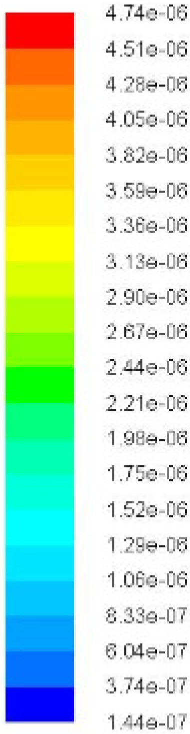
Table 1: Contour of velocity streamline.

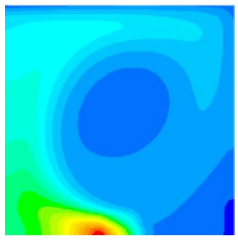
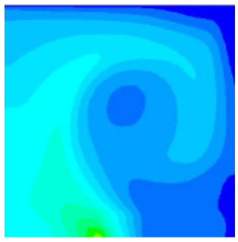
Reynolds number	Velocity streamline	Reynolds number	Velocity streamline
9000		19200	
14200		30700	

Pollutant Dispersion. Development of clusters of buildings or skyscrapers in major cities with extensive transportation system in narrow streets is one of the reasons for the deterioration of air quality. Most of the researchers shows that airflow over different canyon configurations gives a very different behaviour as opposed to the free stream wind flow (Sini et al. [8]) and it often directly affects the comfort of its inhabitants. Improvements can be made if the effective natural ventilation with acceptable degree of turbulence inside the canyons is achieved. Pollutant dispersion is strongly dependent on the structure of the urban boundary layer and its interactions with the rural boundary layer and the synoptic flow. Pollutant is harmful to the human being. Normally the type of pollutant is carbon monoxide that produces from the car exhaust. As a developer, they should provide a necessary of air ventilation to ensure the health of the human. Table 2 shows the contour of carbon monoxide mass fraction inside a street canyon with different Reynolds numbers. One significant findings from this study shows the effective dilution of pollutant at higher Reynolds number. The dilution of pollutant is becoming more effective at higher Reynolds number due to the increase of velocity within the street canyon, which helps to dispersed more pollutant out of the canyon via rooftop near the leeward wall in accordance with the rotational direction of vortex.

Table 2: Contour of Carbon Monoxide Mass Fraction for different Reynolds numbers.

<table>
<tr><th>Carbon monoxide mass fraction</th><th>Reynolds number</th><th>Carbon monoxide mass fraction contour</th></tr>
<tr><td rowspan="4">



</td><td>9000</td><td></td></tr>
<tr><td>14200</td><td></td></tr>
<tr><td>19200</td><td></td></tr>
<tr><td>30700</td><td></td></tr>
</table>

Based on the CO mass concentration graphs as shown Figure 3 at Re = 9000, the pollutant concentration is generally higher along the leeward wall compared to windward wall. At higher Reynolds number, which are at Re = 14200 (Figure 4), Re = 19200 (Figure 5) and Re

Penerbit Akademia Baru
Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 8, No. 1. Pages 11-19, 2015

= 30700 (Figure 6) considered in this study, similar trends of high pollutant concentration along the leeward wall compared to windward wall as at Re = 9000 were observed. Moreover, the pollutant concentration is higher near the bottom and reducing along the height leeward wall. The results of pollutant dispersion obtained in this study, in which higher pollutant concentration along the leeward wall compared to windward wall is supported by numerous previous studies such as Meroney et al. [9]and Liu et al. [10].

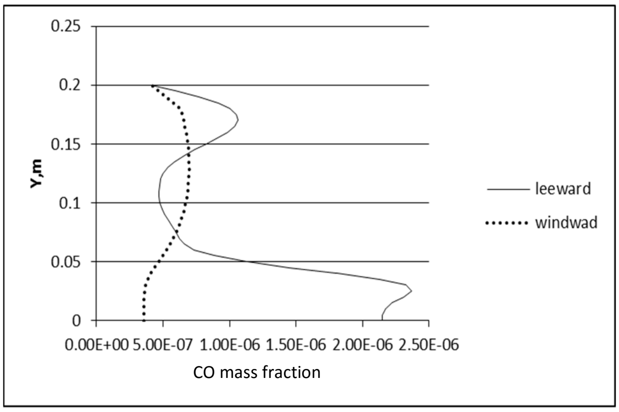


Figure 3: Graph of Y against CO mass fraction (Re=9000).

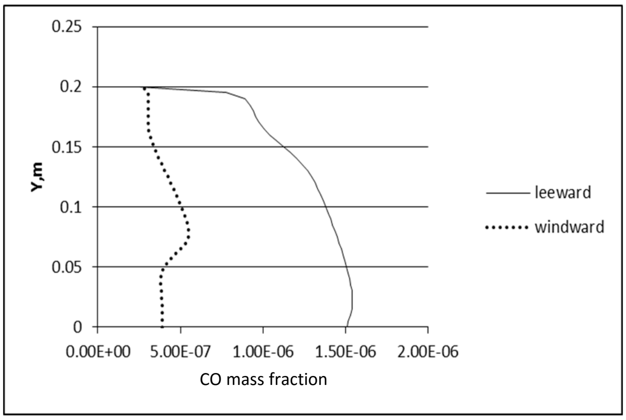


Figure 4: Graph of Y against CO mass fraction(Re=14200).

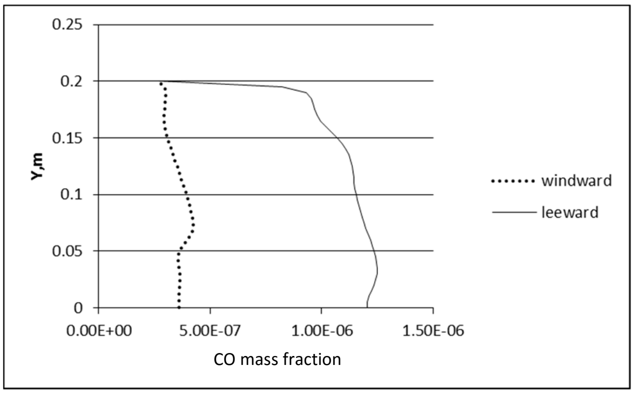


Figure 5: Graph of Y against CO mass fraction(Re=19200).

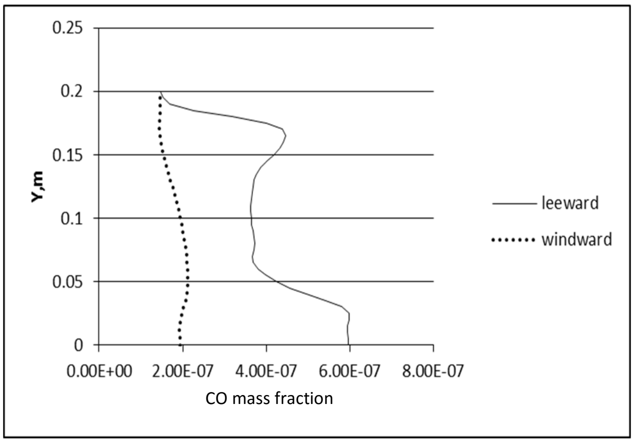


Figure 6: Graph of Y against CO mass fraction (Re=30700)

5.0 CONCLUSION

In this study, mesh independence study has been identified to simulate wind flow and pollutant dispersion in a street canyon. Next, the effect of different Reynolds numbers on flow structure and pollutant dispersion was further investigated using the predetermined mesh. LES was found to yields accurate result which is close to wind tunnel experiment by Allegrini et al. [5]. For the effect of Reynolds number to the flow structure, it was found that only one vortex is formed at the middle of the street canyon. The formation of vortex remain unchanged even at higher Reynolds number. It was also found that as the velocity is increase, the vortex is intensify. Different Reynolds numbers were observed to have affected the pollutant pattern inside the canyon. Generally leeward wall has higher pollutant concentration as compared with windward wall. The higher Reynolds number will provide better air ventilation within the street canyon. This is as the result of increased in vortex intensity which acts as the transport medium for pollutants. This study did not assess the effect of pollutant concentration towards the risk by human through respiratory inhalation but rather provides an insight from the fluid mechanics perspective towards the possible enhancement of air ventilation within a street canyon by an increase in freestream wind speed, which in turn promote better air quality.

REFERENCES

[1] A.W.M. Yazid, N.A.C. Sidik, S.M. Salim, K.M. Saqr, A review on the flow structure and pollutant dispersion in urban street canyons for urban planning strategies, Simulation: Transaction of the Society for Modeling and Simulation International 90 (2014) 892-916.

[2] S.E. Nicholson, A pollution model for street-level air, Atmospheric Environment 9 (1967) 19-31.

[3] J. Smagorinsky, General Circulation Experiments with the Primitive Equations. I. The Basic Experiment, Monthly Weather Review 91 (1963) 99-164.

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 8, No. 1. Pages 11-19, 2015

[4] A. Walton, A.Y.S. Cheng, Large-eddy simulation of pollution dispersion in an urban street canyon—Part II: idealised canyon simulation, Atmospheric Environment 36 (2002) 3615-3627.

[5] J. Allegrini, V. Dorer, J. Carmeliet, Wind tunnel measurement of buoyant flows in street canyons, Building and Environment 35 (2012) 234-256.

[6] T.R. Oke, Street design and urban canopy layer climate, Energy and Buildings 11 (1988) 103-113.

[7] H. Huang, Y. Akutsu, M. Arai, M. Tamura, A two-dimensional air quality model in an urban street canyon: Evaluation and sensitivity analysis, Atmospheric Environment 34 (2000) 689-698.

[8] J.F. Sini, S. Anquetin, P.G. Mestayer, Pollutant dispersion and thermal effects in urban street canyons, Atmospheric Environment 30 (1996) 2659-2677.

[9] R.N. Meroney, M. Pavageau, S. Rafailidis, M. Schatzmann, Study of line source characteristics for 2-D physical modelling of pollutant dispersion in street canyons, Journal of Wind Engineering and Industrial Aerodynamics 62 (1996) 37-56.

[10] C.H. Liu, D.Y.C. Leung, M.C. Barth, On the prediction of air and pollutant exchange rates in street canyons of different aspect ratios using large-eddy simulation, Atmospheric Environment 39 (2005) 1567-1574.