Modelling of Flow Structure and Pollutant Dispersion in Symmetric Street Canyon

Nur Hamizah binti Mohamad Yusoff\textsuperscript{a}, and Nor Azwadi Che Sidik\textsuperscript{b}

\textsuperscript{a}Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor Bahru
\textsuperscript{b}nhamizah91@gmail.com, azwadi@fkm.utm.my,

\textbf{Keywords}: Symmetric street canyon, k-\(\varepsilon\) model, flow structure, pollutant dispersion, FLUENT.

\textbf{Abstract}. The quality of air condition is one of the major factors affecting the health of people living in urban areas. Different geometry of street canyon coupled with different wind direction will result in various concentration of pollutant accumulated in the canyon. The purpose of this research is to study the flow structure and also pollutant dispersion in a 3-D symmetric street canyon using computational fluid dynamics. Different k-\(\varepsilon\) turbulent models are used to simulate the flow structure and the result is compared to experimental. The effects of aspect ratio and Reynolds Number to the flow structure and pollutant dispersion in a street canyon are investigated. The studied canyons are avenue, regular and deep canyon with aspect ratio of 0.4, 1 and 2 respectively and Reynolds Numbers of 9000 and 30700.

\textbf{Introduction}

Flow structure and particle dispersion inside urban street canyon has always been a significant scope to study due to its diverse aspects and practical purposes. Nicholson \cite{8} describes as the relatively narrow street in between buildings that line up continuously along both sides. Hunter et al. \cite{7} stated that the canyon structure has been the center of attention for researchers as different variables such as geometry, wind profile and city climate will produce distinct environment inside the canyon itself. One of the significant properties of street canyon is the flow pattern inside the canyon created by the movement of the wind on rooftop level.

Tall buildings that surrounded an urban street canyon limit the ventilation of air inside and when it coupled with heavy traffics that emit pollutant, it will create a poor air quality condition that has become a major concern in urban area. The dispersion of pollutants mostly depends on the flow profile or air inside the canyon. Over the past two decades, numerous researches had been done to investigate the flow profile inside a street canyon through various methods such as field observation, fluid experiments using wind tunnels and computational fluid dynamics simulation. Based on the researches done, it is found out that flow regime, mean flow and turbulence statistics, dispersion mechanism, thermal effect on flow and dispersion and bulk effects of buildings on mean flow and turbulent kinetic energy are important aspects in understanding the flow structure and pollutant dispersion in an urban street canyon.

Flow inside a street canyon has been investigated numerically and experimentally for decades. Over the year, vast improvements are made in this area with the advancement of technology such as CFD software. It made the process of simulating a flow faster and easier. Previous studies on wind flow field and pollutant dispersion inside street canyons are mostly done in two dimensional. There are few three dimensional cases but they did not compare the result of different k-\(\varepsilon\) to experimental results.

This study will use three dimensional street canyon model and the results from simulation will be validated against a wind tunnel experiment. Three different turbulent model namely Standard, Renormalization Group (RNG) and Realizable k-\(\varepsilon\) will be used to simulate and the results will be compared to determine the best k-\(\varepsilon\) model for simulating flow structure and pollutant dispersion in a three dimensional symmetric street canyon. Building aspect ratio and Reynolds number will be the manipulating variables in the simulations. The pollutant will be assumed massless and non-reactive.
and the source is from vehicular emission in the middle of street canyon only. Results from this research will show the effect of different aspect ratio and wind velocity to the flow pattern and the pollutant concentration on the building inside the canyon. It will also compare the difference between 2-D and 3-D modelling on street canyon features.

**Geometry Modelling**

The three-dimensional model of street canyon used in this simulation is as shown in Figure 1 (View along the canyon; front view) by using Design Modeler in ANSYS FLUENT. The model is based on previous wind tunnel measurement done by Allegrini et al. [1]. Some modifications are made to reduce the computational time without influencing the result.

![Fig. 1 Dimension of Model](image)

For variation of aspect ratio, the height is maintained but the width is adjusted. In case of AR=1, H=W=2m, AR=0.4, W=0.5m and for AR=2, W=0.1m. The dimension for pollutant is 0.1x0.2x0.6m.

**Experimental Setup**

ANSYS FLUENT software is used to solve the numerical calculation. The geometry is meshed using hexahedral and equidistant mesh. The number of cells is increased for mesh independent study purpose only in x and y direction. The mesh size in z direction remained the same for all cases.

![Fig. 2 Geometry Modelling](image)

Boundary conditions are set accordingly. Wall type is set to be stationary and have no slip shear condition. The roughness constant is fixed to a constant of 0.5. Flow rate rating for outflow is fixed as 1. For inlet type in the other hand, external data is needed. The data of boundary layer velocities, kinetic energy and turbulent dissipation for different Reynolds number are extracted from wind tunnel measurement by Allegrini et al. [1]. These data are then saved as profile type and extracted to FLUENT. Boundary condition for outlet is set as outflow.
In order to include the pollutant, species transport is switched on to define the air-carbon monoxide mixture properties. In cell zone condition, carbon monoxide part is set to have a source term of 8.3333 kg/m$^3$s. This value is obtained from 1 g/s calculated from the formula by Tsai and Chen [10]:

$$q_{ik}(t) = \frac{EF_{ik}(t) \times N_k(t)}{A_k \times 1000}$$

(1)

where $EF_{ik}$ is the emission factor of pollutant i and $N_k$ is the average traffic flow rate; subscript k refers to the kth lane. $N_k$ is determined from measurements so $q_{ik}$ can be evaluated once $EF_{ik}$ is known. The emission factor of CO is given by Taneeb [9] was used in this calculation. The situation is assumed to be slow moving car (30 m/s) moving along a 60 m canyon. In an hour average time, 4500 cars are assumed to have passed through the canyon.

Results & Discussion

Validation

The simulation is validated against wind tunnel measurement by Allegrini et al. [1]. In order to find the most suitable turbulence model, the simulation is done in different k-\(\varepsilon\) models namely Standard, RNG and Realizable. The results are then plotted against result from the wind tunnel measurement. Figure 3 shows the result of Y (Situated in the middle of the canyon) against x-velocity for different turbulent model (AR=1, AR=9000).

From the graph, it is clear that RNG k-\(\varepsilon\) model is the best for modelling fluid flow over a cavity such as wind flow over wind canyon. This is in line with result obtained from numerical study by Baik and Kim [2, 3] where RNG k-\(\varepsilon\) turbulence model was shown to improve upon the standard k-\(\varepsilon\) model in simulating turbulent kinetic energy field near the upwind edge of the building. RNG k-\(\varepsilon\) also has been proven by Chan et al. [5] to be the best model for simulating 2-D street canyon.

Flow Structure

The flow structure depends on the velocity of air flowing at roof level and also the geometry of the canyon. Figure 4 shows velocity contour of different Reynolds number and aspect ratio inside the street canyon.
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.

In low wind speed and AR=1, one circular shaped vortex formed in the middle of the canyon. As the aspect ratio decrease to 0.4, the vortex stretched horizontally and its centre shift to the right. But in deep canyon (AR=2), two vortices are spotted aligned vertically. The primary vortex situated near the roof level with the centre at the middle is much stronger compared to the secondary vortex situated near the backwind building. All of the said vortices are moving in clock-wise direction except for the secondary small vortex which flow in the opposite.

As the wind speed increase, the flow structure inside the street canyon for aspect ratio 1 and 0.4 maintained the same but for AR=2, the secondary vortex will merge with the primary vortex to form only one vertically stretched vortex which flow in clock-wise direction. Stretched vortex is caused by the increasing vorticity in the stretched direction but it is also limited by the control volume. The fact that vorticity increase with component velocity explains the formation of the flow structures as stated above.

When the roof level velocity flow across a canyon increase, the velocity variation across the canyon will also increase, thus creating vortex with high strength. From the velocity contour, it shows that higher building aspect ratio will result in lower air velocity at pedestrian level but higher Reynolds number will have the opposite effect. This is due to the distance between the roofs, where the free-stream velocity is to the ground.

Viewing from building aspect, the ratio of wind velocity at backwind building to upwind building decreased with the reduction of aspect ratio and Reynolds number. This can be explained by the fact that when air flow into a street canyon, it will strike first the backwind building than the ground and upwind building therefore losing momentum along the way due to friction or any obstructions on pedestrian level. The results obtained for AR=1 and AR=0.4 shows that the flow structures are almost the same along the street canyon which mean that the effect of z-axis is nearly non-existent which is the same as two-dimensional simulation done by Baik and Kim [2, 3] and three-dimensional analysis by Christian and Banerjee [4]. For deep canyon case, the z-direction does have a significant effect to the flow field as the flow structure along the canyon keep changing. Length of street canyon should be taken into consideration when modelling in three-dimension.

The analysis of flow structure is important as it will affect the dispersion of pollutant inside a street canyon. Turbulent intensity, vortex shape and strength are major factors that dictate the behavior of pollutant transportation.

**Pollutant Dispersion**

Pollutant emitted from vehicles such as carbon monoxide is very harmful to human being. It is necessary for urban area developer to build street canyon with proper air ventilation as to increase the health of people reside inside it. Figure 6 shows the contour of carbon monoxide mass fraction.
inside a street canyon with different aspect ratio and Reynolds number. For variation shown in Figure 7 and Figure 8, the measurement is taken at leeward and windward side of buildings.

Fig. 6 Contour of Carbon Monoxide mass fraction

Fig. 7 Variation of Molar Concentration along the height of street canyon (Re=9000)

Fig. 8 Variation of Molar Concentration along the height of street canyon (Re=30 700)

From the carbon monoxide contour and the molar concentration variation, leeward wall generally has higher pollutant concentration as compared to windward wall for all aspect ratio and both Reynolds number except for case AR=2 coupled with Re=9000. In this case, high concentration of pollutant resides at the bottom of windward side as opposed to leeward side. The concentration at the windward wall gradually decrease with the height of the building and the concentration is equal for both side at y=0.085 m. The concentration keeps declining until the half of the building height and it became almost constant after that. The distinctive dispersion pattern for this case is affected
by the formation of vertically-aligned double vortex in the street canyon. The primary vortex does not allow vehicular emissions carried by the secondary vortex from the ground level to rise beyond the mid canyon height.

High concentration at leeward wall is the result of formation of wind flow vortex characterized by down draft near the windward building and updraft at the leeward building. The average concentration level in street canyon increases as the building aspect ratio increases. As discussed before, higher aspect ratio will result in formation of two vortices but the vortex nearer to the pollutant source is much weaker than that at the roof level. This weak vortex is caused by the dominance of molecular diffusion over advection and turbulent diffusion therefore resulting in high accumulation of pollutant at the bottom of the street.

Figure 7 and figure 8 also show that the pollutant concentration decrease along the height of the leeward side of backwind building which is consistent with the findings from 2-D simulation by Huang et al. [6] and street measurement by Xie et al. [11]. But the declination is tremendous as the speed increase. This phenomenon is contributed by the increase of vortex strength with the increasing free-stream velocity. This vortex facilitates the ventilation of vehicular emission through roof level. But for the windward side of the upwind building, the pollutant concentration shows little to no change along the height of building.

From the carbon monoxide mass fraction contour, it can be observed that higher Reynolds number enhanced the removal of pollutant from inside the canyon. Referring to Figure 4.9, the pattern shows that the pollutant concentration is almost halved when the Reynolds number increase. This trend can be explained in term of turbulent intensity where the wind speed affecting the vortex strength causing the increment in the vertical mean velocity which improve the transportation of pollutants to the roof level.

![Figure 9](image)

**Fig. 9 Variation of Molar Concentration along the height of street canyon (AR=0.4)**

Similar to flow structure in street canyon, the results obtained for pollutant dispersion shows good agreement on the effect of z-axis. This effect is not tangible in AR=1 and AR=0.4 but is significant in the case of AR=2 for both Reynolds number. Pollutant accumulates at the middle of the street canyon at high Reynolds number but dispersed quite evenly at certain length along the canyon for low Reynolds number. Street canyon’s length may be one of the factors that influence study of street canyon in 3-D modelling.

**Conclusion**

From the comparison against wind tunnel experiment, RNG k-Ɛ turbulent model shows the best result for simulating 3D symmetric street canyon which is also in line with finding by Chan et al. (2002) where RNG k-Ɛ is the best turbulence model for simulating 2D street canyon.

For the effect of building aspect ratio to the flow structure, it is found out that only one vortex will formed inside street canyon of AR=1 and AR=2 but there is two vortices formed in AR=2 in low aspect ratio. But when the Reynolds number increase, the pattern and number of vortex
remained the same for case AR=1 and AR=0.4 but for AR=2, the two vortices will merged to form one vertically-stretched vortex. It is also found out that higher velocity will increase the turbulent intensity inside a canyon thus increasing the vortex strength.

Different aspect ratio and Reynolds number will also affect the dispersion pattern of pollutant inside the canyon. Leeward wall generally has higher pollutant concentration as compared to windward wall except for the case of AR=2 where the concentration is higher at the windward wall up until a certain height of the building. This result discrepancy is caused by the formation of secondary weak vortex near the ground. Higher aspect ratio will result in higher pollutant entrainment in the canyon due to the increase in distance between the pollutant source and free-stream velocity. For the effect of Reynolds number, higher Reynolds number will provide better ventilation for the street. This is as the result of increasing vorticity which acts as the transport medium for pollutants.

Considering the length of street canyon, the dispersion of pollutant is even along the street canyon for AR=1 and AR=0.4 but AR=0.2 shows the highest pollutant concentration is at the middle of street for both Reynolds number. Same pattern is encountered with the flow structure where the effect of z-axis is nearly non-existent for AR=1 and AR=0.4 but is significant in AR=2.

References