DEVELOPMENT AND VALIDATION OF A COMPUTATIONAL FLUID DYNAMICS MODELLING METHODOLOGY FOR ISOLATED AND URBAN STREET CANYON CONFIGURATIONS USING WIND TUNNEL MEASUREMENTS

MADHAVAN VASUDEVAN[1], BIDROHA BASU[1,2], FRANCESCO PILLA[2] & AONGHUS MCNABOLA[1,3]
[1] Department of Civil, Structural & Environmental Engineering, Trinity College, Dublin, Ireland.
[2] School of Architecture Planning and Environmental Policy, University College Dublin, Ireland.
[3] Global Centre for Clean Air Research, Department of Civil and Environmental Engineering, Faculty of Engineering and Physical Sciences (FEPS), University of Surrey, United Kingdom.

ABSTRACT

Precise prediction of air quality in a street canyon under diverse conditions could be established through the comprehensive validation of velocity of wind profiles and the concentration distribution of pollutants. In this study, a two-step approach was developed using Computational Fluid Dynamics simulations. The first step involved the validation of wind velocity profiles obtained using wind tunnel experimental measurements of an isolated street canyon discussed in ref. [1], while the second step focused on the validation of dispersion of pollutants from wind tunnel measurements discussed in ref. [2] conducted on isolated and urban street canyons. The wind velocity profiles obtained at five distinct vertical planes between the leeward and windward walls in the wind tunnel study [1] were validated by simulating the 2D cross-section of the entire wind tunnel domain with high accuracies; R^2 values of 0.931–0.986 were obtained across the canyon depth. The concentration distribution of the pollutant in the wind tunnel study [2] were validated for a range of velocities (0.5, 1, 2 and 4 m/s) using both 2D and 3D models. A verification of the Reynolds independent nature of the flow was performed by comparing the wind tunnel and street scale models and suitability of employing K-ε turbulence model with Enhanced Wall Treatment and K-ε Low Reynolds Number Model for the wind tunnel scale, and Standard Wall Functions for the street scale were observed. A 2D simulation of urban street canyon flow representing the whole wind tunnel cross-section in the flow direction was also studied to observe repetitive flow nature and thereby a potential to employ fully developed flow conditions for the same. The urban street canyon flow is established through the means of fully developed periodic flow profiles, which inherently restricts the additional mass sources in the flow domain. The emission scenario in the fully developed flow was captured by means of flow profile mapping at the upwind edge of the leeward building. To estimate the minimum number of downwind canyons required to keep up the fully developed flow profile at the target street canyon, a parameterization of the same was performed. Finally, the validation of the concentration profiles was obtained with parameterization of the Schmidt number, and an optimal Schmidt number was obtained in the case of using Realizable K-ε turbulence model. The developed and validated methodology provides a robust and efficient means of modelling air pollution dispersion in the isolated and urban street canyons for future research investigations.

Keywords: 2D and 3D simulations, CFD, fully developed flows, isolated street canyon, urban street canyon, validation.

1 INTRODUCTION

Air pollution was estimated to be responsible for 6.4 million premature deaths worldwide in 2015 and is expected to cause up to 9 million deaths annually by 2060 if there is a failure to

ISSN: 2046-0546 (paper format), ISSN: 2046-0554 (online), http://www.witpress.com/journals
DOI: 10.2495/CMEM-V10-N2-104-116

implement aggressive control measures [3]. Monitoring and modelling form integral parts of mitigation measures especially for the performance analyses of source receptor pathway structures including low boundary walls [4, 5] and vegetation [6] to mention a few. In modelling, validations are generally performed against field and wind tunnel measurements. Wind tunnel studies offer the benefit of providing concentration measurements of dispersion in street canyons under fixed and repeatable conditions [7]. The wind tunnel studies pertaining to micrometeorological environments have included analyses on flow around an isolated building [8], flow across an isolated [1, 2, 9, 10], non-isolated [7, 10] and urban street canyons [2, 11], modifications to standard building geometries [12, 13], built source receptor pathway intervention inclusions [14], vegetation inclusions [15, 16], complex 3D flows involved through gaps in street canyons [17], buoyant flows [9] and different roadway configurations [18].

Kovar-Panskus *et al.* [19] compared flow in isolated street canyons including aspect ratios (w/H) from 0.3 to 2 by validating the turbulent flow quantities. Allegrini *et al.* [9] conducted the validation using 2D RANS models to mimic the buoyant flows observed in wind tunnel measurements including an isolated street canyon with different heating scenarios and found a minimal influence of y+ values on the prediction of dominant flow patterns developed by different wall functions and Low Reynolds Number Models (LRNM). Chan *et al.* [20] validated the flow across the isolated street canyon modelled in ref. [2] using Standard, Realizable and RNG K-ε turbulence models in 2D and concluded that the RNG K-ε turbulence model was the most suitable for the application as it provided the least variation in the non-dimensional concentration co-efficient, K for different reference wind velocities. Takano and Moonen [21] analyzed the influence of different angles of pitched roofs in urban environment using fully developed flow conditions and validated the results against the urban street canyon flow modelled in ref. [2].

As ref. [1] includes a comprehensive measurement of velocity profiles inside and over the roof of the canyon and measurements in ref. [2] provide an opportunity to evaluate the dispersion of emissions with similar framework of conditions for both isolated and urban street configurations; the former and latter have been utilized for velocity and concentration profile validations, respectively. In contrast to the typical urban street canyons modelled in the past, a specific plausible requirement for the installation of a source–receptor intervention structure in a street canyon where the flow otherwise without the structure is fully developed is considered. Also, in verifying Reynolds' independent nature of the flow, suitability of LRNMs and wall functions were analyzed for both street and wind tunnel scales in the context of computational effectiveness and accuracy.

2 OPEN STREET CONFIGURATION

The sections to follow are organized such that the isolated street canyon flow validations including velocity and concentration prediction models are categorized separately (current section) from the concentration prediction model for the urban street canyon flow validation (Section 2.1).

2.1 Methodology–velocity validation

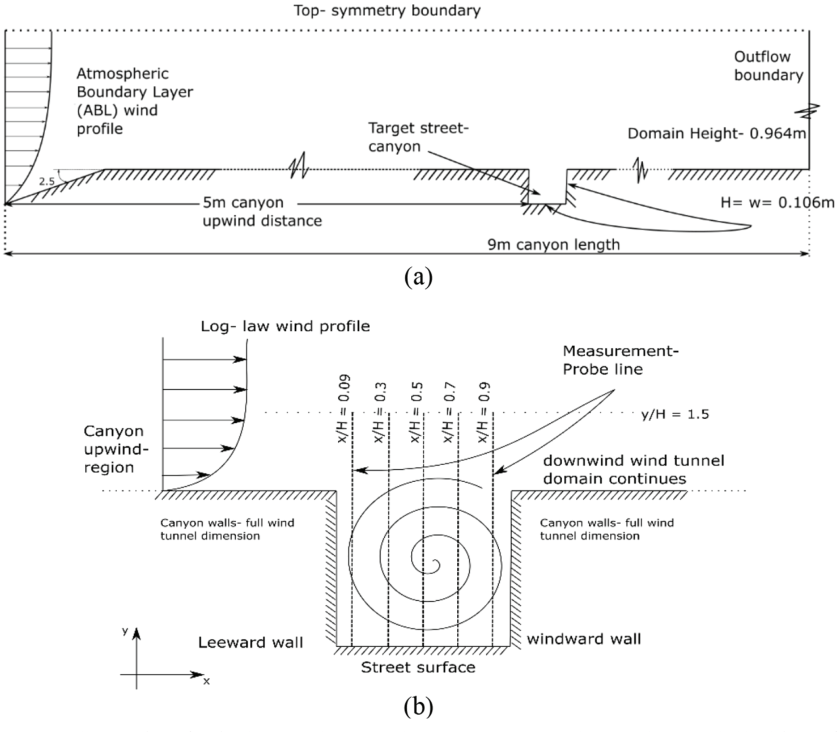
Figure 1 schematically represents the longitudinal cross-section of the symmetric isolated street canyon with an aspect ratio of 1 (w = H = 0.106 m) modelled in ref. [1] and highlights the vertical planes including x/H = 0.09, 0.3, 0.5, 0.7 and 0.9, respectively, from the leeward wall where the streamwise velocity values were probed in the measurement and so in the current Computational Fluid Dynamics (CFD) model for validation. The reference free stream velocity considered in the experiment was 8 m/s. More details on the experimental setup can be obtained from ref. [1]. The inlet velocity was defined using a User Defined Function (UDF) to represent the log law wind profile as described in eqn (1).

$$\mathrm{u} = u_{*} \ln\left(\frac{y}{\delta}\right) \tag{1}$$

Here, u is the velocity of the approaching wind at a height y above the ground. u_{*} is the frictional velocity, and δ is the thickness of boundary layer. The turbulence parameters of the inlet are specified using turbulence intensity and turbulence length scales based on the measurements. As the k-ε model is considered sufficiently as the representative for conditions including flow separation only because of flow over sharp edges, the validation study was carried out using RNG k-ε and Realizable k-ε turbulence models in ANSYS-FLUENT.

2.2 Methodology–concentration validation

Figure 2 schematically represents the computational domain considered in the current CFD model and highlights the locations along the leeward and windward walls where K was probed in the measurement and in the CFD model for validation. The reference free stream velocity values included in ref. [2] were 0.5, 1, 2, 3, 4 and 5m/s. The lateral injections of



(a)

(b)

Figure 1: Schematic of wind tunnel (a) and measurement planes (b) modelled in ref. [1].

pollutants in the experiments were physically modelled using 302 thin holes of 0.25 mm diameter, placed continuously along the street canyon to represent a line source of emission. The overall emissions from the holes accounted for 100 l/h of air and 4 l/h of ethane gas. More details of the experimental measurements could be obtained from ref. [2]. The inlet velocity, TKE and Turbulent Dissipation Rate (TDR) were defined using UDFs to represent the atmospheric boundary layer (ABL) wind profile as described in eqn (2a–c).

$$\mathrm{u} = u_{ref}\left(\frac{y-d_0}{\delta-d_0}\right)^{\alpha} (a) \quad \mathrm{k} = \frac{u_*^2}{\sqrt{c_\mu}} (b) \quad \varepsilon = \frac{u_*^3}{k(y+y_0)} (c) \tag{2a–c}$$

Here, the constants u_{ref}, d_0, α, y_0, u_* and κ represent the mean flow velocity, displacement height of the flow, power law coefficient of the ABL, aerodynamic roughness height of the ABL, frictional velocity and the Von-Karman constant. $c_\mu = 0.09$, and u, k and ε represent the velocity, TKE and TDR.

The species transport model in ANSYS-FLUENT was employed to evaluate the dispersion of the ethane gas. The lateral injection of the ethane gas was facilitated by assigning selected zones in the computational domain for the creation of the emission gases and prescribing horizontal momentum to the same based on mass and momentum flux balancing.

The height and width of the building and the width of the street separating them were 0.06 m and 20 m each in the wind tunnel and street scale studies, respectively. The meshing strategy is largely dependent on the near wall treatment approach adopted in the model. Considering the two different scales, Standard Wall Function (SWF) for RNG k-ε turbulence model, LRNM k-ε turbulence model and Enhanced Wall Treatment (EWT) for Realizable and RNG k-ε turbulence models were considered in the current study. The SWF provides a computationally efficient solution by modelling the near wall region as the first cell from the wall is in the logarithmic region, typically in the y^+ range between 30 and 300. On the other hand, the LRNMs are computationally demanding as the first cell from the wall needs to be in the viscous sublayer resolving in the range of $y^+ < 5$. EWT is a blended near wall formulation that is compatible with both SWF and LRNM meshes. A similar y^+ resolved, say $y^+ = 30$ for a reference velocity of 1 m/s, the first cell height estimate for wind tunnel and street scale would be $\Delta y_{wt} = 0.0069$ m and $\Delta y_{fs} = 0.01$ m, respectively. This requirement is especially demanding for the 3D models. Apart from highlighting the increased computational requirement for using LRNM for street scale simulations, it also shows practical constraints in using the SWF for wind tunnel scale simulations. $\Delta y_{wt} = 0.0069$ m being larger than $0.1H_b$ potentially violates the guideline prescribed in ref. [22] that recommends for at least 10 grid cells along the height of the building surface to resolve the mean flows of the problem comprehensively. This problem with SWF is felt stronger for further smaller reference velocities. On the other hand, SWF provides the opportunity to control surface roughness. As aerodynamically rough flows exceeding critical building Reynolds number $\left(\mathrm{Re_H} = \frac{uH}{\nu}\right)$ is expected to exhibit similar flow and thereby dispersion characteristics for different length scales as long as all the geometric features in the domain are simultaneously scaled up/down equally and the boundary conditions are applied identically, the results from the two scales were verified for Reynolds number independence. Therefore, results using SWF for street scale were referenced with results obtained from EWT and LRNM for the wind tunnel scale along with measurements to setup a basis for further street scale dispersion investigations.

3 URBAN STRET CONFIGURATION

3.1 Reference measurements for the validation of urban street canyons flows

The urban street canyon modelling in ref. [2] utilized a similar physical modelling procedure as in the open street canyon scenarios to inject the emission gases homogeneously along the entire span of the street canyon. Urban roughness was introduced by the inclusion of 20 bars (building representations) upstream and 8 bars downstream to the target street canyon. The scenario including a reference velocity of 2 m/s and an aspect ratio of 1 is considered in for validation. The velocity profile of the wind is represented by the same characteristic equation as for the open street configuration (eqn (2)), but the parameters including displacement height (d_0) and exponential co-efficient α were considered as 0.057 and 0.20, as in the measurements. Both the height of the buildings and street width correspond to 0.06 m, like the open street canyon case. Further details into the experimental procedure could be obtained from ref. [2].

3.2 Reference measurements for validation of urban street canyons flows

Unlike the isolated street canyons that are seldom encountered by urban street flows, several cities include parallel street canyons. Such configurations could be effectively captured by the modelling of repetitive flow profiles. The fully developed flow profiles obtained using periodic boundary conditions are useful to represent the repetitive nature although it corresponds to accounting an infinite number of similar canyons both upstream and downstream to the target street canyons. The methodology implemented in the current study addresses the applicability of using the fully developed flows for the street canyons in cities with a finite number of parallel streets and circumventing following natural limitations that prohibits one from using it extensively for studying air pollution in urban environments.

First, the 2D cross-section of the wind tunnel flow including all the 28 building structures was simulated to realize the repetitive nature of the flow. The street canyons were classified to exhibit fully developed flow profiles after seven to eight street canyons approximately [2]. Figure 3 shows a plot of TKE and velocity magnitude at the 10th, 15th, 20th and 25th buildings' upwind edge. The high level of agreement of velocity and minimum loss of TKE at subsequent downwind building locations signify the close-to-approaching fully developed

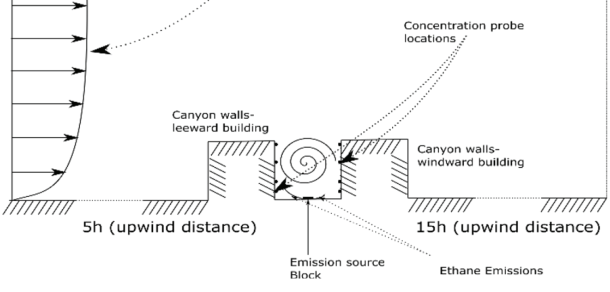


Figure 2: Schematic diagram representing the computational domain, incoming velocity profile and ethane gas concentration measurement probing locations in ref. [2].

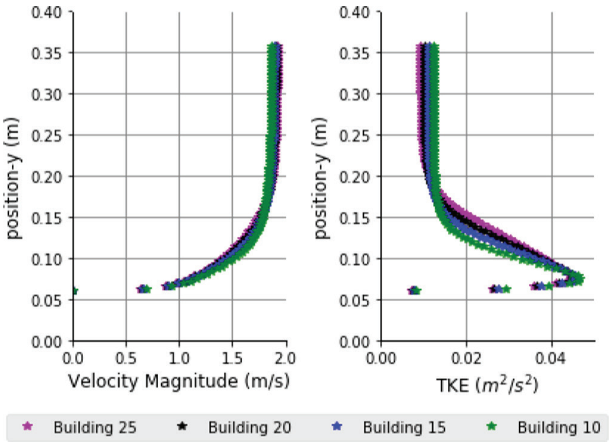


Figure 3: Plots representing the flow over upwind edge of 10th, 15th, 20th and 25th buildings from simulation performed on whole wind tunnel setup: Velocity and TKE.

nature of the flow, also justifying the application of periodic flows to represent more realistic finite number of parallel street canyons as well and not just theoretically accurate assumption of infinite number of street canyons. Second, the fully developed flow models require a constant mass flow rate across the periodic boundaries and thereby prohibits the addition of pollutants in this case. In ref. [21], passive scalar transport method was adopted for evaluating the dispersion of ethane gas. Finally, inclusion of intervention structures in fully developed flows signify the inclusion of the structure in all the parallel streets but such structures are typically installed phase-by-phase in reality because of its cost, installation time and need for its performance evaluation. Therefore, an approach to systematically evaluate the intervention's performance in a phased manner is desirable and time efficient from a computational standpoint when considering fully developed flow models. Latter two concerns have been addressed by a flow profile mapping method in the current study. This is a common CFD technique used in many applications to prescribe the inlet with an already evaluated flow profile at that location. Using this method here, as shown in Fig. 4, the flow field is first solved for the fully developed case without pollutants and the flow field variables are mapped on to the inlet of the simulation considering the pollutant emissions.

Unlike in ref. [21], where the periodic flow is established through pressure drop across the periodic boundaries, mass flow rate is evaluated here through eqn (3) and is applied in the current study.

$$\dot{m} = \int_{H}^{6H} \rho u_{ref} \left(\frac{y - d_0}{\delta - d_0} \right)^{\alpha} b dy \tag{3}$$

Here ρ is the density of air, b is the width of the flow and other parameters represent the same as in open street canyon case. For reference free stream velocity of 2 m/s, the mass flow rate across the periodic domain was computed to be 0.585 kg/s. The geometry and boundary conditions adopted for the establishment of fully developed flows is schematically represented in Fig. 4. The computational domain includes a width of 2H as shown in Fig. 4. First, to ensure if the domain completely represents the periodicity of the flow as performed in ref. [21], the velocity, TKE and TDR profiles at the upwind edge of the building were compared

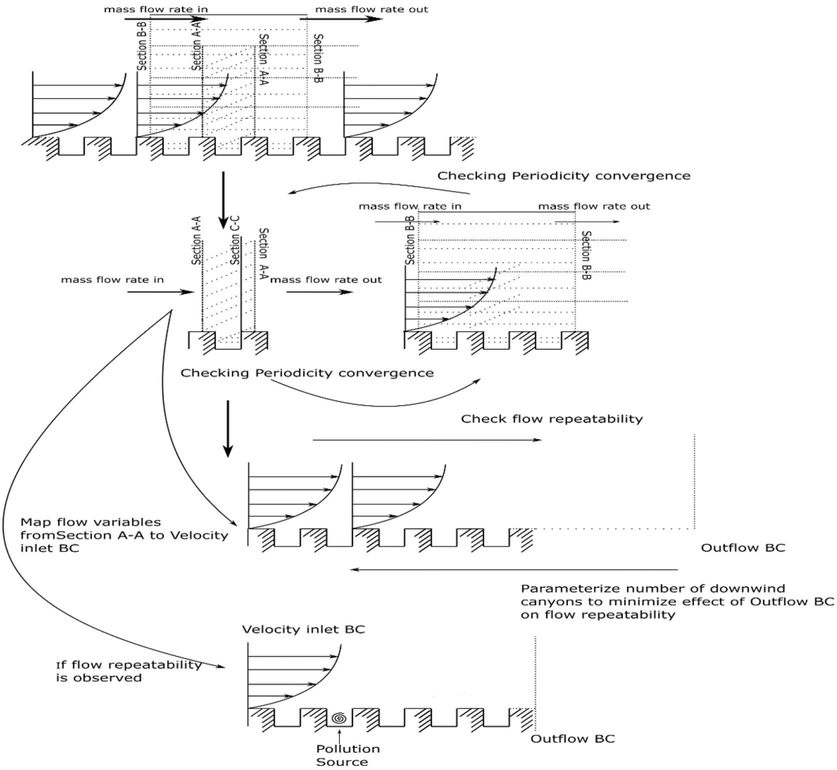


Figure 4: Schematic representation of using fully developed flows for urban street canyon flows modelled in ref. [2].

to corresponding observations from a similar model including a computational domain width of 6H (comparison not include considering brevity).

This confirms that the selected region of study and mass flow rate used are representative of the fully developed nature of the flow. Next, as shown in Fig. 4, the profiles of velocity components, TKE and TDR from Section C-C are mapped to the inlet of the model including flow through finite number of canyons bounded by an outflow boundary condition at the downwind end of the domain. (Henceforth, the CFD model considering the fully developed flow and model evaluating the pollutant dispersion with the use of mapped flow properties would be referred as fully developed flow and mapped models.)

To quantify the impact of outflow boundary condition and to ensure it does not drastically influence the periodicity of the flow across the target street canyon, parameterization of number of downwind street canyons was performed. This included mapped models involving downwind domain width from 4H to 15H. The choice of the maximum downstream distance was based on the experimental setup in ref. [2], where eight buildings positioned downstream of pollution source corresponded to 15H.

Figure 5 shows the comparison of TKE at x = 0.36 m in the mapped models from two different simulations that included three canyons (total domain width of 0.36 m) and nine canyons (total domain width of 1.08 m), respectively. Additionally, TKE at the inlet from model with three canyons and TKE at the outlet from simulation with nine canyons is

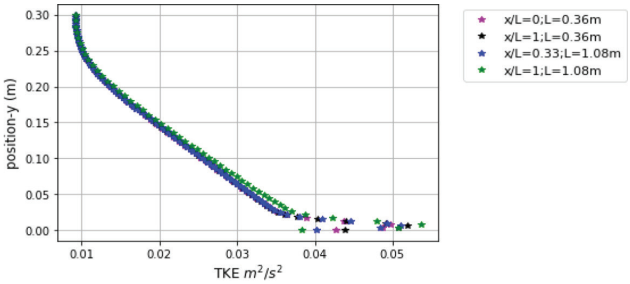


Figure 5: Parameterization of number of downwind canyons in the mapped model.

included to illustrate the minimal loss in TKE and validity to assume fully developed flow across any number of downwind canyons. Figure 5 highlights the two important aspects; not only is the reduction of TKE from inlet to a distance of three canyons across minimal, but also the effect of outflow boundary condition in disturbing the periodic nature of the flow (once initialized with a fully developed flow) is also minimal. This could be realized by minimal difference in TKE at x = 0.36 m for both the considerations including domain widths of 0.36 m and 1.08 m.

Considering brevity, results from assuming other downwind length is not included in Fig. 5. For the benefit of representing the downstream domain as in the experiments, eight buildings downstream to the pollution source was considered for further analysis. The mapped model included modelling of pollution sources like the isolated street canyon model and Realizable and RNG k-ε turbulence models using EWT were similarly employed for the analyses.

4 RESULTS

4.1 Open street canyon

4.1.1 Velocity validation

The streamwise velocity validation at five distinct vertical planes in the street canyon along with experimental measurements are shown in Fig. 6. Figure 6 shows that both RNG k-ε and Realizable k-ε models predicted the velocities at different vertical planes comprehensively in agreement with ref. [1]. The characteristic trend of a steep streamwise velocity increase at the roof level signifying the clockwise vortex existing in the isolated canyon is captured by both the turbulence models. The R^2 values obtained between the CFD model with RNG k-ε turbulence model and measurements at the five planes including x/H = 0.09, 0.3, 0.5, 0.7 and 0.9 were 0.931, 0.953, 0.984, 0.986 and 0.944, respectively.

4.1.2 Concentration validation

Figure 7 shows the comparison of concentration prediction for the isolated street canyon case modelled in ref. [2]. First, the 2D and 3D RNG k-ε turbulence models using EWT (red and green lines in Fig. 7) were compared for all the velocities, and the agreement assured the representativeness of the 2D analyses for the pollution dispersion scenario. Considering the robustness of EWT in handling the meshes corresponding with first cells in viscous, buffer and logarithmic layers, the grids for each velocity providing mesh converged results with EWT were used for analyses with SWF and LRNM for the wind tunnel scale.

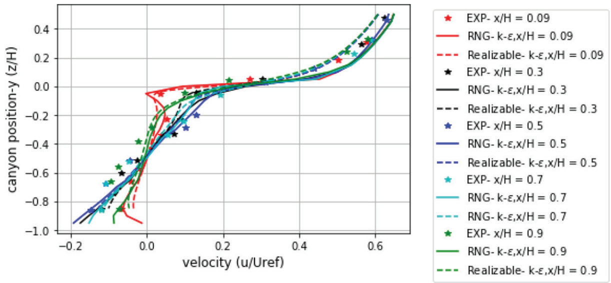


Figure 6: Comparison of numerical velocity predictions by RNG k-ε and Realizable k-ε models with experimental measurements in street canyon modelled in ref. [1].

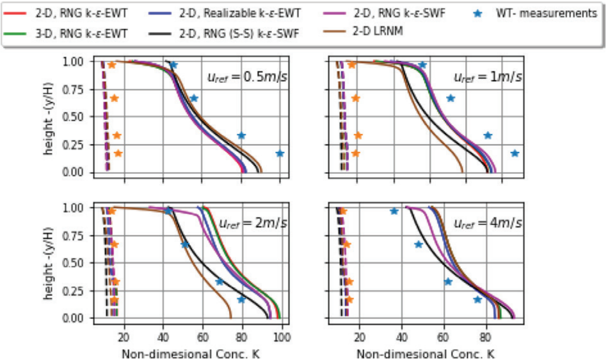


Figure 7: Comparison of wall functions and turbulence models at wind tunnel and street scales with experimental measurements obtained for dispersion modelled in ref. [2]. (S-S in the legend stand for street scale and continuous and dashed lines represent the leeward and windward sides). Velocities: 0.5, 1, 2 and 4 m/s.

Noticeably, all the models underpredicted the concentration of ethane gas along the windward wall for lower velocities. The 2D RNG k-ε and Realizable k-ε turbulence models using EWT (red and blue lines in Fig. 7) provided similar results unlike in ref. [20]. They over-estimated the concentration of ethane gas along leeward wall in the near-roof regions, especially for higher velocities of 2 and 4 m/s. These over-predictions were reduced in models employing SWF (magenta line in Fig. 7) and LRNM (brown line in Fig. 7). However, because models including SWF for wind tunnel scale resulted in extremely low y^+ values than the general guidelines, further fine mesh resolutions resulted in unphysical solution thereby highlighting the unsuitability of SWF for the scenario. LRNM on the other hand provided very good agreement with the trends observed in the measurement except for its over prediction of near-roof concentration along the leeward wall for reference velocity of 4 m/s. Finally, the street scale simulations using SWF (black line in Fig. 8) provided the best agreement of all with the measurements. This observation could be attributed to using the SWF for its most appropriate scenario where the first cells from the wall lie in the logarithmic region. Unlike in the case of wind tunnel scale, the street scale simulations resulted in good mesh independence with SWF. However, considering the appropriateness of the mesh for the near wall treatment, the results using LRNM and EWT for the wind tunnel scale were expected to provide better agreement than using SWF for street scale as their near wall region were more resolved. The agreement of the street scale model using SWF with the experimental measurements are numerically quantified in Fig. 8.

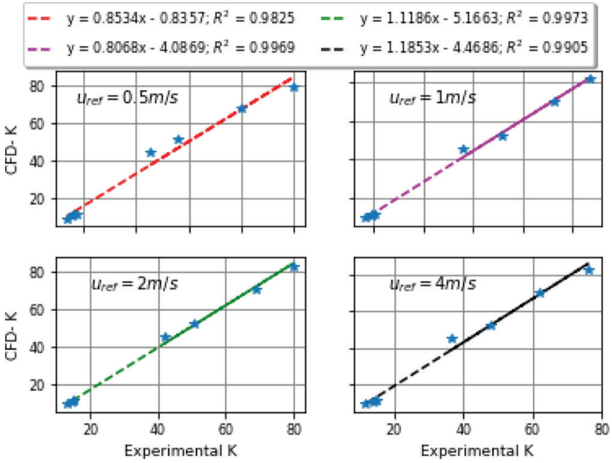


Figure 8: Numerical comparison of street scale concentration predictions using RNG k-ε-SWF and measurements in ref. [2] for velocities including 0.5, 1, 2 and 4 m/s.

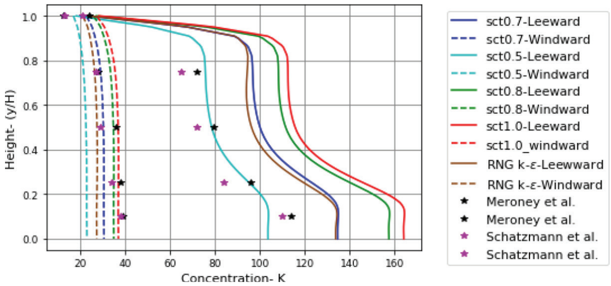


Figure 9: Validation of urban street canyon flows referred in ref. [2] using 2D Realizable (for a range of Schmidt numbers) and RNG k-ε turbulence models in the mapped model.

4.2 Urban street canyon

Considering the representativeness of 2D models, performance of Realizable and RNG k-ε turbulence models and the Reynolds' independent nature of the flow observed in isolated street canyon flows, validation of the urban street canyon flows is considered at the wind tunnel scale using 2D RNG and Realizable k-ε models with EWT. Figure 9 shows that the leeward side agrees well with measurements for lower Schmidt values while the windward side agrees well for the higher Schmidt values. Therefore, default Schmidt of 0.7 is recommended in the case of using Realizable k-ε model. RNG k-ε is also seen to result in similar concentration prediction as by default Schmidt number in Realizable k-ε and thereby assures the validity of using both these turbulence models for the urban street canyon flow applications.

5 CONCLUSIONS

In this study, a robust two-step validation procedure has been established to ensure that the concentration and velocity profiles are predicted for studies from two different wind tunnel studies. The selection of the two wind tunnel studies for the validation has also been established.

Suitability of using LRNM k-ε, EWT with both Realizable and RNG k-ε in the case of wind tunnel scale modelling and SWF for street scale modelling were established and thereby also a validation for Reynolds' independent nature of the flow in the case of open street canyon. For urban street canyons, a clear emphasis on the requirement to consider fully developed flows in the special case to address target street canyon for inclusion of passive interventions in future and potential to employ a flow mapping technique has been stated using Realizable and RNG k-ε turbulence models. The validation procedure not only provides a basis for systematic evaluation of the influence of passive interventions in urban street canyons to be carried out in a phase-by-phase manner but also a direct comparison with the open street canyon configuration.

ACKNOWLEDGEMENTS

The authors would like to acknowledge that this study was part funded by the Irish Environmental Protection Agency's Research Programme 2014–2020 through Project Grant Number 2019-PhD-AF-6. The EPA Research Programme is a Government of Ireland initiative funded by the Department of Communications, Climate Action and Environment.

REFERENCES

[1] Kovar-Panskus, A., Louka, P., Sini, J.F., Savory, E., Czech, M., Abdelqari, A., Mestayer, P.G. & Toy, N., Influence of geometry on the mean flow within urban street canyons–a comparison of wind tunnel experiments and numerical simulations. *Water, Air and Soil Pollution: Focus*, **2(5)**, pp. 365–380, 2002. https://doi.org/10.1007/978-94-010-0312-4_26

[2] Meroney, R.N., Pavageau, M., Rafailidis, S. & Schatzmann, M., Study of line source characteristics for 2-D physical modelling of pollutant dispersion in street canyons. *Journal of Wind Engineering and Industrial Aerodynamics*, **62(1)**, pp. 37–56, 1996. https://doi.org/10.1016/s0167-6105(96)00057-8

[3] Landrigan, P.J., Fuller, R., Acosta, N.J., Adeyi, O., Arnold, R., Baldé, A.B., Bertollini, R., Bose-O'Reilly, S., Boufford, J.I., Breysse, P.N. & Chiles. T., The Lancet Commission on pollution and health. *The Lancet*, **391(10119)**, pp. 462–512, 2018. https://doi.org/10.1016/s0140-6736(17)32345-0

[4] McNabola, A., Broderick, B.M. & Gill. L.W., A numerical investigation of the impact of low boundary walls on pedestrian exposure to air pollutants in urban street canyons. *Science of the Total Environment,* **407(2)**, pp. 760–769, 2009. https://doi.org/10.1016/j.scitotenv.2008.09.036

[5] Gallagher, J., Gill, L.W. & McNabola, A., Optimizing the use of on-street car parking system as a passive control of air pollution exposure in street canyons by large eddy simulation. *Atmospheric Environment*, **45(9)**, pp. 1684–1694, 2011. https://doi.org/10.1016/j.atmosenv.2010.12.059

[6] Gromke, C., Buccolieri, R., Di Sabatino, S. & Ruck, B., Dispersion study in a street canyon with tree planting by means of wind tunnel and numerical investigations–evaluation of CFD data with experimental data. *Atmospheric Environment*, **42(37)**, pp. 8640–8650, 2008. https://doi.org/10.1016/j.atmosenv.2008.08.019

[7] Kastner-Klein, P. & Plate, E.J., Wind-tunnel study of concentration fields in street canyons. *Atmospheric Environment*, **33(24–25)**, pp. 3973–3979, 1999. https://doi.org/10.1016/s1352-2310(99)00139-9

[8] Mirzai, M.H., Harvey, J.K. & Jones, C.D., Wind tunnel investigation of dispersion of pollutants due to wind flow around a small building. *Atmospheric Environment*, **28(11)**, pp. 1819–1826, 1994. https://doi.org/10.1016/1352-2310(94)90321-2

[9] Allegrini, J., Dorer, V. & Carmeliet, J., Buoyant flows in street canyons: validation of CFD simulations with wind tunnel measurements. *Building and Environment*, **72**, pp. 63–74, 2014. https://doi.org/10.1016/j.buildenv.2013.10.021

[10] Sagrado, A.P., van Beeck, J., Rambaud, P. & Olivari, D., Numerical and experimental modelling of pollutant dispersion in a street canyon. *Journal of Wind Engineering and Industrial Aerodynamics*, **90(4–5)**, pp. 321–339, 2002. https://doi.org/10.1016/s0167-6105(01)00215-x

[11] Chang, C.H. & Meroney, R.N., Concentration and flow distributions in urban street canyons: wind tunnel and computational data. *Journal of Wind Engineering and Industrial Aerodynamics*, **91(9)**, pp. 1141–1154, 2003. https://doi.org/10.1016/s0167-6105(03)00056-4

[12] Meroney, R.N., Leitl, B.M., Rafailidis, S. & Schatzmann, M., Wind-tunnel and numerical modeling of flow and dispersion about several building shapes. *Journal of Wind Engineering and Industrial Aerodynamics*, **81(1–3)**, pp. 333–345, 1999. https://doi.org/10.1016/s0167-6105(99)00028-8

[13] Llaguno-Munitxa, M., Bou-Zeid, E. & Hultmark, M., The influence of building geometry on street canyon air flow: validation of large eddy simulations against wind tunnel experiments. *Journal of Wind Engineering and Industrial Aerodynamics*, **165**, pp. 115–130, 2017. https://doi.org/10.1016/j.jweia.2017.03.007

[14] Ding, S., Huang, Y., Cui, P., Wu, J., Li, M. & Liu, D., Impact of viaduct on flow reversion and pollutant dispersion in 2D urban street canyon with different roof shapes-Numerical simulation and wind tunnel experiment. *Science of the Total Environment*, **671**, pp. 976–991, 2019. https://doi.org/10.1016/j.scitotenv.2019.03.391

[15] Gromke, C. & Ruck, B., Influence of trees on the dispersion of pollutants in an urban street canyon—experimental investigation of the flow and concentration field. *Atmospheric Environment*, **41(16)**, pp. 3287–3302, 2007. https://doi.org/10.1016/j.atmosenv.2006.12.043

[16] Gromke, C., A vegetation modeling concept for building and environmental aerodynamics wind tunnel tests and its application in pollutant dispersion studies. *Environmental Pollution*, **159(8–9)**, pp. 2094–2099, 2011. https://doi.org/10.1016/j.envpol.2010.11.012

[17] Cui, P.Y., Li, Z. & Tao, W.Q., Investigation of Re-independence of turbulent flow and pollutant dispersion in urban street canyon using numerical wind tunnel (NWT) models. *International Journal of Heat and Mass Transfer*, **79**, pp. 176–188, 2014. https://doi.org/10.1016/j.ijheatmasstransfer.2014.07.096

[18] Heist, D.K., Perry, S.G. & Brixey, L.A., A wind tunnel study of the effect of roadway configurations on the dispersion of traffic-related pollution. *Atmospheric Environment*, **43(32)**, pp. 5101–5111, 2009. https://doi.org/10.1016/j.atmosenv.2009.06.034

[19] Kovar-Panskus, A, Louka, P., Mestayer, P.G., Savory, E., Sini, J.F. & Toy, N., Influence of geometry on the flow and turbulence characteristics within urban street canyons–Intercomparison of wind tunnel experiments and numerical simulations. In *Proceedings of the 3rd International Conference on Urban Air Quality* (pp. 19–23), 2001.

[20] Chan, T.L., Dong, G., Leung, C.W., Cheung, C.S. & Hung, W.T., Validation of a two-dimensional pollutant dispersion model in an isolated street canyon. *Atmospheric environment*, **36(5)**, pp. 861–872, 2002. https://doi.org/10.1016/j.atmosenv.2009.06.034

[21] Takano, Y. & Moonen, P., On the influence of roof shape on flow and dispersion in an urban street canyon. *Journal of Wind Engineering and Industrial Aerodynamics*, **123**, pp. 107–120, 2013. https://doi.org/10.1016/j.jweia.2013.10.006

[22] Franke, J., *Best practice guideline for the CFD simulation of flows in the urban environment*. Meteorological Inst., 2007.