CWE-8
June 7-11, 2026
London, ON, Canada

A comparative analysis of wall-function roughness models for atmospheric boundary layer simulations

A. Piazza[a,b], A. Ricci[c], G. T. Bitsuamlak[d], M. Repetto[e], M. Burlando[f]

[a,e f]University of Genoa, Genoa, Italy,
[b,d]Western University, London, Ontario, Canada,
[c]IUSS Pavia, Pavia, Italy,
[a]*alessia.piazza@edu.unige.it*, [b]*apiazz2@uwo.ca*, [c]*alessio.ricci@iusspavia.it*, [e]*gbitsuam@uwo.ca*
[d]*maria.pia.repetto@unige.it* and [f]*massimiliano.burlando@unige.it*

SUMMARY

The growing demand for full-scale Computational Fluid Dynamics (CFD) simulations of urban environments has led to an increasing need for improved accuracy in Reynolds-Averaged Navier–Stokes (RANS) modelling, a widely used approach due to its low computational cost and demonstrated reliability. This approach relies on wall functions to capture the near-wall flow behaviour; however, despite extensive prior studies, several uncertainties persist. In particular, two main categories of roughness-based wall functions can be identified: one derived from the Nikuradse (1933) sand-grain experiments and another based on the aerodynamic roughness length z_0. In this study, the performance of these two wall-function formulations is evaluated using both open-source and commercial CFD software, and the results are compared against wind-tunnel data to assess their differences and the implications for urban-flow simulations.

Keywords: *RANS, ABL, wall function, TKE, Urban flow*

1. INTRODUCTION

Computational Fluid Dynamics (CFD) simulations are widely employed in the wind engineering community to assess, for example, pedestrian-level wind conditions, urban microclimates, and pollutant dispersion. Traditionally, such simulations have been conducted at model scale, consistent with the scales typically used in wind-tunnel experiments, partly due to the substantial computational cost associated with full-scale simulations. However, with recent advancements in High Performance Computing (HPC), full-scale CFD simulations have gained increasing attention in recent years. Among the three primary turbulence-modelling approaches, Direct Numerical Simulation (DNS), Reynolds-Averaged Navier–Stokes (RANS), and Large Eddy Simulation (LES), not all are feasible for full-scale urban applications. DNS offers the highest fidelity but remains prohibitively expensive at realistic Reynolds numbers. LES provides a good representation of unsteady flow structures but can still be too computationally demanding at neighborhood or city scales. LES is also increasingly being mandated for atmospheric boundary layer (ABL) flows in wind load evaluations, as reflected in documents such as the upcoming CWE pre-standard developed by ASCE/NIST. However, RANS models, by contrast, remain the most widely used approach for full-scale urban flow investigations due to their relatively low computational cost and demonstrated reliability, particularly for modelling mean-flow-dominated phenomena. This turbulence modelling approach relies on the use of wall-functions to capture the behavior of the flow near the wall. In the past years wall functions have been object of investigations by Blocken et al., (2007), Hargreaves and Wright (2007), Yang et al., (2009), where the limitations of RANS in simulating neutral Atmospheric Boundary Layer (ABL) with the commercial CFD codes have been documented.

Most studies must rely on implicit modeling of wall roughness. The foundation of rough-wall modelling is based on the experiment carried out by Nikuradse (1933), where the equivalent sandgrain roughness k_s (also called roughness height) was estimated based on the experimental data of turbulent flow in fully rough regimes. In another widely used study, Hargreaves and Wright (2007) proposed an alternative implementation based on aerodynamic roughness length z_0 with application for ABL simulation. The aim of this study is to compare the performance of these two wall functions in full-scale simulations, with particular focus on the velocity and Turbulent Kinetic Energy (TKE) profiles. Particular attention will also be given to surface roughness, and mesh refinement influence the performance of the wall functions. While most previous studies have analyzed the behavior of wall functions exclusively using the standard k–ε turbulence model, a wider range of models (e.g., realizable k–ε, k–ω SST) will be investigated in the present study, owing to their common use urban flow simulation. The results obtained from both open-source and commercial software will be compared and validated against wind-tunnel profiles, with the ultimate objective of providing practical guidelines for users when selecting and applying wall functions in urban flow simulations using RANS.

2. THEORETICAL BACKGROUND

Finite-volume discretization of the momentum equations introduces a momentum sink term representing the wall shear stress for the wall-adjacent cells. This term accounts for the momentum exchange between the wall and the fluid. While the solver computes the velocity at the first cell center and the wall velocity is known from the boundary condition, the wall shear stress must still be estimated to close the system of equations. CFD solvers use linear interpolation across cell centers to obtain a pseudo continuous solution, therefore if the mesh is sufficiently fine, the wall shear stress can be directly computed from the first cell center, i.e. $\frac{\tau_{wall}}{\rho} = \nu \frac{du}{dz} = \nu \frac{U_p}{z_p}$, where $U_p$is the solver-computed velocity at the first cell center, z_p. However, for most practical engineering applications, the first cell center lies far from the viscous sublayer, and a linear approximation of the gradient produces an incorrect wall shear stress because of the strong velocity gradient very close to the wall.

Wall functions address this issue by modeling the near-wall behavior using the universal law-of-the-wall. For smooth walls, the law-of-the-wall gives $u^+ = z^+$ for $z^+ < 5$ and $u^+ = \frac{1}{\kappa}\ln(Ez^+)$ for $z^+ > 30$, where the logarithmic form represents the log-layer and E the wall roughness parameter. Therefore, when the mesh is fine enough that the first cell center lies inside the viscous sublayer, a linear profile to compute the gradient can be directly used-this is called the "wall-resolved" approach. However, when the first cell center lies in the logarithmic region ($z^+ > 30$), the velocity gradient may still be approximated using a linear profile if the near-wall effective viscosity ν_t is adjusted so that $(\nu + \nu_t)\frac{U_p}{z_p}$ is consistent with the wall shear stress implied by the log-law. This adjustment of the near-wall viscosity is the essential mechanism behind all wall-function formulations.

For rough walls, the logarithmic relation is modified. A common form based on Nikuradse's sand-grain roughness is $u^+ = \frac{1}{\kappa}\ln\left(\frac{Ez^+}{1+C_s k_s^+}\right)$, while in wind engineering applications, a formulation based on aerodynamic roughness length often used is $u^+ = \frac{1}{\kappa}\ln\left(\frac{z^+ + z_0^+}{z_0^+}\right)$. These lead to two families of wall functions. In sand-grain-based models, the adjusted near-wall viscosity becomes

$\nu_t = \frac{u^* z_p}{\frac{1}{\kappa}\ln\left(\frac{E z^+}{1+C_S k_S^+}\right)}$, whereas in the aerodynamic roughness-length based formulation $\nu_t = \frac{u^* z_p}{\frac{1}{\kappa}\ln\left(\frac{z^+ + z_0^+}{z_0^+}\right)}$. Most commercial CFD solvers implement the traditional sand-grain formulation. In OpenFOAM, the nutkRoughWallFunction uses the sand-grain formulation, whereas atmNutkWallFunction employs the aerodynamic roughness-length formulation. In principle, aside from small implementation details, both should give similar results when a consistent mapping between k_s and z_0 is used.

3. DOMAIN AND BOUNDARY CONDITIONS

The computational domain used for this study is a large-scale domain characterized by 5000 m (L) x 100 m (W) x 500 m (H) dimensions (Figure 1). The domain is discretized with a structured, uniform grid in the longitudinal and transversal directions. In the vertical direction, the grid is stretched using a geometric progression to position the centroid of the first wall adjacent cell at a height of 0.5 m.

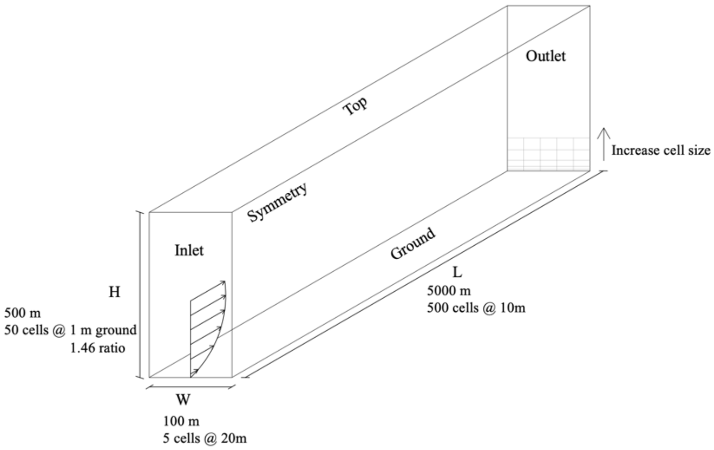

Figure 1. Computational domain and boundary conditions applied for this study.

At the inlet, boundary profiles of wind speed, turbulent kinetic energy, and dissipation rate are prescribed following the methodology of Richards and Hoxey (1993). As an initial configuration, the *k–ε* turbulence model is employed. The reference wind speed is set to 10 m/s at a height of 6 m, and the corresponding values of $u*$ and z_0 are adjusted for each test case. A pressure inlet–outlet boundary condition is imposed at the outlet. At the domain top, a fixed shear-stress condition is applied in accordance with Richards and Hoxey (1993), while at the ground surface the two wall functions introduced in Section 2 are implemented. The first set of simulations are carried out using the open-source software OpenFOAM.

4. OUTCOMES

The results from three test cases (z_0 = 0.01 m, z_0 = 0.3 m, and z_0 = 0.5 m) are presented in Figure 2. As shown, the wind-speed and TKE profiles extracted at x = 2500 m exhibit noticeable discrepancies, even with identical inlet conditions and solver settings for each case. For the wind speed in particular, the percentage difference between the two wall functions increases with roughness length, reaching 46% at a height of 1.5 m for the case with z_0 = 0.5 m. This difference in wind speed is reflected in the turbulent kinetic energy profile: while the two TKE profiles overlap for the lowest-roughness case (z_0 = 0.01 m), increasing roughness leads to divergences up to 1.5 m²/s² for the roughest case. These differences can affect the evaluation of urban flow for pedestrian comfort, thermal comfort and pollutant dispersion.

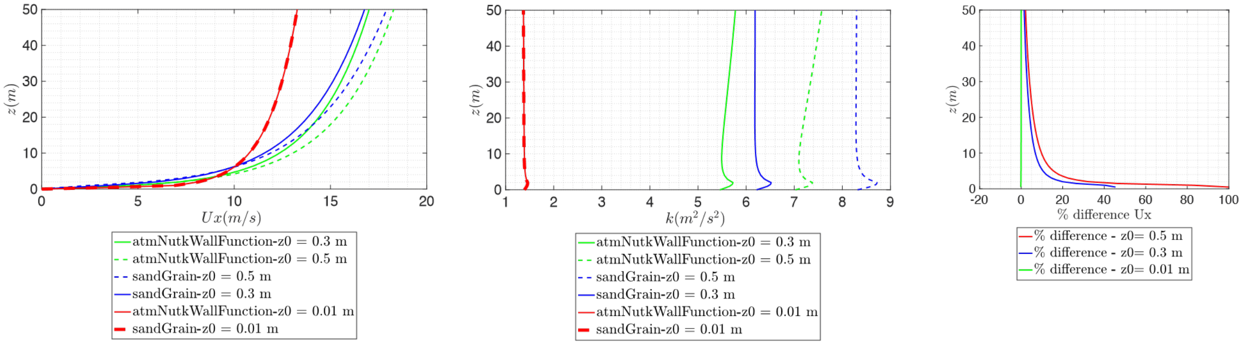

Figure 2. ABL profiles extracted at x = 2500 m of the Ux component of wind speed (left), TKE (centre) and percentage difference (right) between the wind speed profiles for the three test cases of roughness.

5. CONCLUSIONS

The results indicate that predictions of wind speed and turbulent kinetic energy are strongly influenced by the choice of the wall function, with discrepancies increasing with wall roughness. These findings demonstrate the importance of carefully selecting wall functions and grid resolution, with possible implications for accurate urban flow modeling.

REFERENCES

Blocken, B., Stathopoulos, T., Carmeliet, J., 2007. CFD simulation of the atmospheric boundary layer: wall function problems. Atmos Environ 41, 238–252. https://doi.org/10.1016/j.atmosenv.2006.08.019

Hargreaves, D.M., Wright, N.G., 2007. On the use of the k-ε model in commercial CFD software to model the neutral atmospheric boundary layer. Journal of Wind Engineering and Industrial Aerodynamics 95, 355–369. https://doi.org/10.1016/j.jweia.2006.08.002

Nikuradse, J., 1933. Laws of Flow in Rough Pipes. VDI-Forschungsheft 361.

Richards, P.J., Hoxey, R.P., 1993. Appropriate boundary conditions for computational wind engineering models using the k-E turbulence model, Journal of Wind Engineering and Industrial Aerodynamics.

Yang, Y., Gu, M., Chen, S., Jin, X., 2009. New inflow boundary conditions for modelling the neutral equilibrium atmospheric boundary layer in computational wind engineering. Journal of Wind Engineering and Industrial Aerodynamics 97, 88–95. https://doi.org/10.1016/j.jweia.2008.12.001