sciendo
Slovak Journal of Civil Engineering
Vol. 33, 2025, No. 2, 64–71
DOI: 10.2478/sjce-2025-0013

COMPARISON OF c_{pe} VALUES FROM CFD AND WIND TUNNEL TESTING FOR VARIOUS SURFACE ROUGHNESSES

Samuel CRUZ [1*], Milan PALKO[1]

Abstract

This paper presents a comparative analysis of the pressure coefficient (c_{pe}) values obtained through wind tunnel experiments and CFD simulations using two approaches for modeling surface roughness i.e, geometrically explicit roughness and equivalent roughness constants in ANSYS Fluent. Five levels of surface roughness were analyzed to assess whether simplified roughness constants can sufficiently approximate geometrically modeled surfaces. The results show close agreement for most cases, which indicates the potential for computational simplification.

Address

[1] Dept. of Building Construction, Faculty of Civil Engineering STU Bratislava, Slovakia

[*] ***Corresponding author:*** samuel.krull@gmail.com

Key words

- c_{pe},
- CFD,
- Roughness modeling,
- Fluent,
- Wind tunnel,
- Turbulence.

1 INTRODUCTION

The accurate prediction of wind effects on buildings and structures is a critical aspect of wind engineering and urban aerodynamics. The external pressure coefficient c_{pe} is a dimensionless number defined as the difference between local pressure and reference static pressure, normalized by dynamic pressure, as per EN 1991-1-4. Pressure coefficients (c_{pe}) on building facades directly affect the estimation of wind loads, natural ventilation, and the dispersion of pollutants. Traditionally, such data has been obtained through experimental testing in Boundary Layer Wind Tunnels (BLWT) (ZHOU, 2024), which provide reliable physical results under controlled conditions. However, these experiments can be time-consuming, expensive, and limited in terms of the variability of parameters.

In recent decades, Computational Fluid Dynamics (CFD) has emerged as a powerful alternative for modeling wind flow around buildings. CFD enables a detailed analysis of flow fields and surface pressures and offers flexibility in geometry, boundary conditions, and flow regimes. Nevertheless, its accuracy heavily depends on how physical surface properties, such as roughness, are modeled. Surface roughness (Smith, 2023) plays a key role in the development of the boundary layer and turbulence characteristics (Yang and Gu, 2016)(He et al., 2015), and its proper representation is essential for replicating real-world conditions.

There are two common approaches to modeling surface roughness in CFD, i.e. geometric modeling, where physical features are explicitly included (Blocken, 2014) in the mesh, and software-defined roughness, where empirical parameters, such as the roughness height (k_s) and roughness constant (C_s), are applied at the boundary conditions. While geometric modeling can offer a higher degree of fidelity, it significantly increases the complexity of the mesh and computational demands. On the other hand, equivalent roughness constants simplify the model but may introduce inaccuracies if not properly calibrated.

Recent studies, such as (Blocken, 2014) ,(Wu and Stathopoulos, 2019), and (Li et al. , 2018), have compared CFD and wind tunnel methods, but limited work directly evaluates both geometric roughness and equivalent sand-grain roughness modeling. This study addresses this gap by systematically comparing the two CFD approaches with experimental data across multiple roughness levels.

This paper presents a comparative study evaluating both approaches against wind tunnel measurements. A modular cuboid model with variable surface roughness was used for experiments in a boundary layer wind tunnel. The same geometry and conditions were replicated in CFD simulations using ANSYS Fluent (Tominaga et al., 2008) (Li et al., 2018) The goals were to assess how closely Fluent's roughness constant method matches the results of the explicit geometric modeling and physical measurements and see whether it could serve as a valid alternative for engineering applications where computational efficiency is crucial.

2 MATERIALS AND METHODS

This study combines numerical simulations using Computational Fluid Dynamics (CFD) and experimental data from wind tunnel testing to evaluate the impact of surface roughness on the pressure coefficient (c_{pe}) of a bluff body. The primary objective is to determine whether simplified roughness constants in CFD can reliably replicate results obtained from geometrically modeled surfaces and physical measurements.

2.1 Numerical Setup in ANSYS Fluent

The atmospheric boundary layer (ABL) was modeled using the logarithmic wind profile implemented through the User-Defined Functions (UDFs) in ANSYS Fluent. The inlet boundary condition was defined by the log-law velocity distribution, which incorporates the aerodynamic roughness length and friction velocity. The turbulent kinetic energy (k) and its dissipation rate (ϵ) were calculated using standard empirical relations, thus ensuring a realistic representation of the development of turbulence.

Two modeling approaches were applied to the CFD simulations:

- **Geometric roughness:** Surface features such as ribs or blocks were physically modeled on the façade, thereby simulating real structural projections like balconies.
- **Equivalent roughness constants:** The surface roughness was simulated through Fluent's wall roughness specification using the sand-grain roughness height (k_s) and roughness constant (C_μ).

Meshing was performed using hexahedral or hybrid cells, with local refinement near the wall boundaries (Raasch and Schroter, 2001). The wall treatment followed the Enhanced Wall Treatment approach, which maintained appropriate y^+ values (typically $30<y^+<300$ or $y^+\approx 1$ depending on the treatment of the turbulence). The Realizable k–ϵ Turbulence Model was selected, due to its suitability for external flows with recirculation zones. The simulations were conducted in a steady-state mode using the SIMPLE algorithm for pressure-velocity coupling.

The vertical profile of the turbulent kinetic energy and wind dissipation rate was calculated using the shear velocity, which incorporates the surface roughness and wind speed at a given height. The inlet boundary condition was defined as a logarithmic profile using user-defined functions (UDF.c), where Equation (1) represents the wind velocity in the x-direction; Equation (2) defines the turbulent kinetic energy; and Equation (3) is the dissipation rate. The logarithmic profile was defined as the inlet parameter as follows:

$$\frac{U}{u^*} = \frac{1}{\kappa} ln\, ln \left(\frac{z+z_0}{z_0}\right), \qquad (1)$$

where u^* is the friction velocity [m/s]; κ is the von Kármán constant [-]; and z_0 is the surface roughness length [m].

$$k = \frac{(u^*)^2}{\sqrt{C_\mu}}, \qquad (2)$$

$$\epsilon = \frac{(u^*)^3}{\kappa(z+z_0)}. \qquad (3)$$

Where k is in [m²/s²] and u^* is in [m/s] and ϵ is in [m²/s³].

2.2 Boundary Conditions

The bottom surface (ground) was defined as a wall with a standard wall function. The top and lateral boundaries were set symmetrically, while the inlet surface was defined according to the logarithmic Atmospheric Boundary Layer (ABL) profile. The roughness height of the bottom surface was set at 0.02 m, which corresponds to an ABL profile for a low-rise urban terrain (a city environment) (Wu and Stathopoulos, 2019).

2.3 The Size of the domain

Pressure coefficients and velocity fields are highly sensitive to the size of a domain. The total dimensions of the computational domain were defined as follows: 8× the building's width on both sides, 8× the building length's in front of the object, 25× behind the object, and 5× the building's height. See Figure 1. As a reference, a cuboid object was considered. A domain of this size ensures that the boundary conditions (the symmetry and outlet) have a minimal impact on the velocity profile. The domain height was chosen to ensure a blockage ratio below 5%, in accordance with recommendations by (Blocken, 2014) and (Franke et al., 2007).

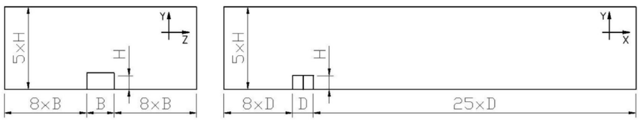

Fig. 1 *Size of the computational domain (Kralik, 2015).*

3 SIMULATION OF A BOUNDARY LAYER IN A CLEAN DOMAIN

3.1 Computational Mesh

The goal was to simulate the domain without the object and compare the results obtained with the actual measured values from the wind tunnel (WT), and to fine-tune the computational parameters to minimize any error. The simulation was performed using the Ansys CFD software suite, including Fluent. Numerous unsuccessful simulations were carried out before achieving an acceptable model. As noted in the section on domain size, the simulation is highly sensitive to the mesh resolution, making it difficult to identify the optimal mesh. The meshing setup allowed elements with a maximum size of 5.53 cm, and additional solver parameters had to be configured to achieve the best possible results. Solver settings included default values for under-relaxation factors for pressure and for momentum, with residuals converged below 1e-5 for all variables.

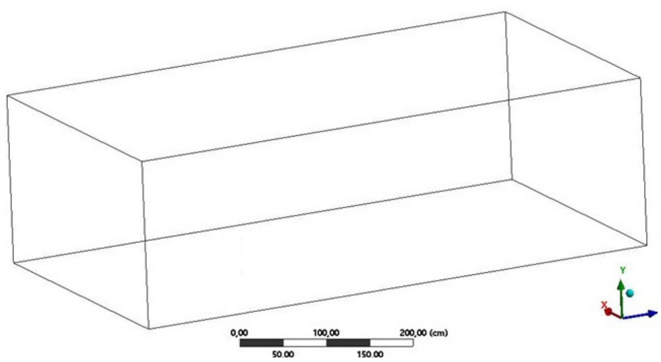

Fig. 2 *Computational domain.*

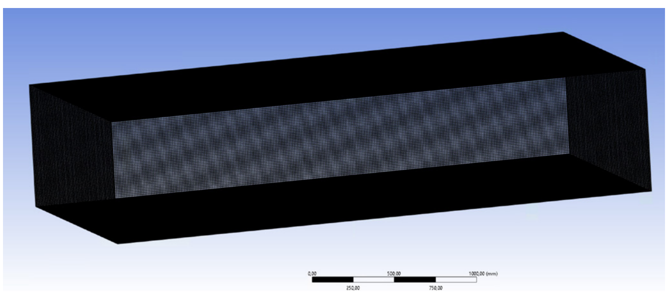

Fig. 3 *Mesh of the computational domain.*

4 NUMERICAL EXPERIMENT WITH THE MODEL

4.1 Geometry and Computational Domain

The numerical simulation was carried out on a model representing a building/block with a defined facade. A 3D CAD model was created for the simulation and subsequently imported into ANSYS Fluent. Two model variants were analyzed i.e. :

- a model with smooth walls (no geometric roughness),
- a model with protrusions representing actual surface features (e.g., balconies, projections).

The computational domain had a cuboid shape with dimensions of X × Y × Z, which ensured that the side and top boundaries did not affect the flow results near the object. The inlet boundary was placed at least 8H (where H is the object's height) upstream from the model, and the outlet boundary was placed at least 18H downstream.

4.2 Meshing and y+

The geometry was meshed using a hexahedral (or hybrid) grid with refined cells near the wall surfaces. To properly apply the wall functions, the y^+ parameter was maintained within the recommended range for the selected turbulence model (typically $30 < y^+ < 300$ for standard wall functions, or $y^+ \approx 1$ when using the Enhanced Wall Treatment). A mesh sensitivity analysis was performed by refining the mesh and comparing the pressure coefficients. Results showed less than 3% variation beyond the chosen mesh density, confirming mesh independence. In our case $y^+ \approx 190$.

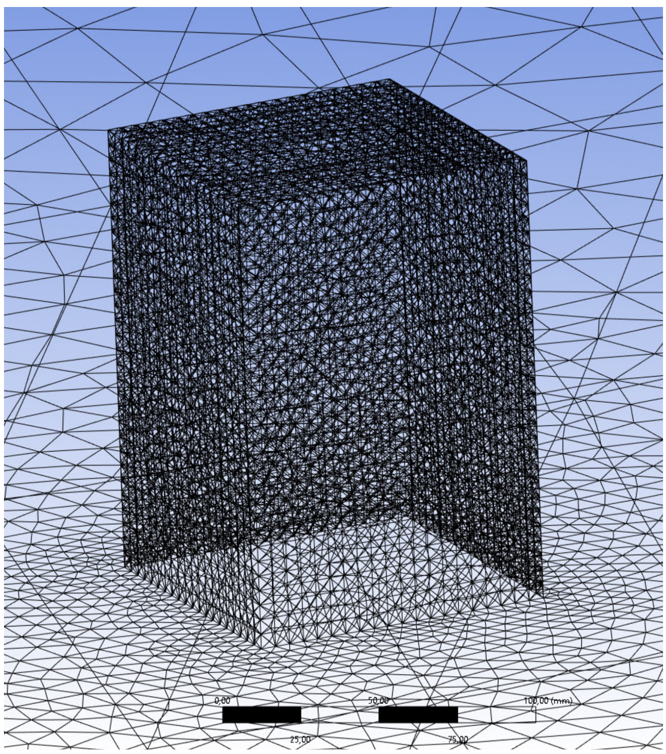

Fig. 4 *Mesh example of a domain.*

 Vol. 33, 2025, No. 2, 64–71

4.3 Turbulence Model and Methods

The Realizable $k - \epsilon$ Turbulence Model with an Enhanced Wall Treatment was selected for the simulation. This model enables a more accurate resolution of near-wall flows, which is essential for evaluating the effect of the surface roughness. The simulations were performed in a steady-state mode using the SIMPLE algorithm for the pressure–velocity coupling.

4.4 Boundary Conditions

- **Inlet:** The wind profile was defined using a logarithmic velocity distribution to consider the surface aerodynamic roughness of the surface as Equation (1).
- **Outlet:** Pressure outlet with a zero gradient.
- **Walls:** Applied either as a geometric roughness (explicit model) or a software-defined roughness using k_s and C_μ values in Fluent.

4.5 Experimental Setup

The wind tunnel experiments were conducted using surface panels with five levels of roughness: 0 mm, 0.5 mm, 2 mm, 3 mm, and 4 mm. Pressure taps measured the c_{pe} values at multiple locations on the surface.

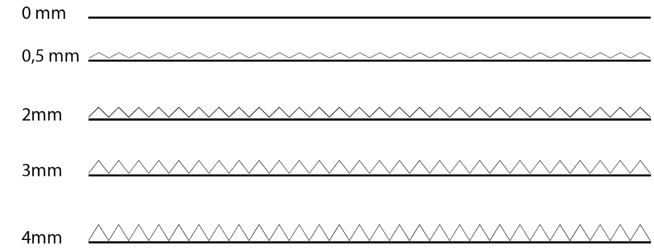

Fig. 5 *Types of geometric wall roughness.*

Fig. 6 *Types of geometric wall roughness.*

4.6 CFD Simulations

The simulations were performed in ANSYS Fluent using two methods:

- **Geometric roughness:** Modeled using detailed pyramidal elements.
- **Roughness constant:** Defined via Fluent's equivalent sand-grain roughness input.

All the simulations used the same turbulence model and boundary conditions for consistency.

5 WIND TUNNEL EXPERIMENT

The experimental measurements were conducted in the Boundary Layer Wind Tunnel (BLWT) at the Slovak Technical University (STU) in Bratislava–Trnávka Slovakia, which is designed to simulate the Atmospheric Boundary Layer (ABL) for scaled models. The tunnel features an open return loop with a 2.35-meter-long working section and an upstream development length of approximately 15 meters.

A modular cuboid model with interchangeable roughness panels (ranging from 0 mm to 4 mm) was manufactured using 3D printing. The model included integrated 1 mm pressure tap holes on all walls and the top surface. Measurement points were distributed at four height levels (bottom, mid-height, near top, and top surface), with 16 evenly spaced taps per level. Each measurement was repeated at least three times for statistical robustness.

A differential pressure scanner (Scanivalve dsa 3217) was used to record steady pressure readings. The mean wind speed was approximately 11.42 m/s at model height, and the turbulence intensity measured using vertical probe arrays was approximately 19.5%, confirming the formation of a fully developed logarithmic wind profile. Artificial roughness elements (e.g., textured plastic sheets or ribbed foils) were applied on the floor to replicate terrain roughness. The same azimuthal wind direction was maintained across all tests to match the simulation setup. The resulting $c_{\square e}$ values were compared with CFD results to evaluate the reliability of both modeling approaches.

5.1 Model and Mounting

Pressure measurements were conducted on a physical cuboid model with dimensions of 100 × 100 mm (base) and 150 mm in height. The model was designed to permit rapid replacement of surfaces with different roughness levels, while maintaining the accuracy and repeatability of the measurements. The base model was 3D-printed with integrated 1 mm diameter holes placed at defined points according to the CFD simulation layout. These holes served as ports for connecting the pressure probe.

Each side of the main model was equipped with pre-fabricated recesses for magnetic attachments, thereby allowing

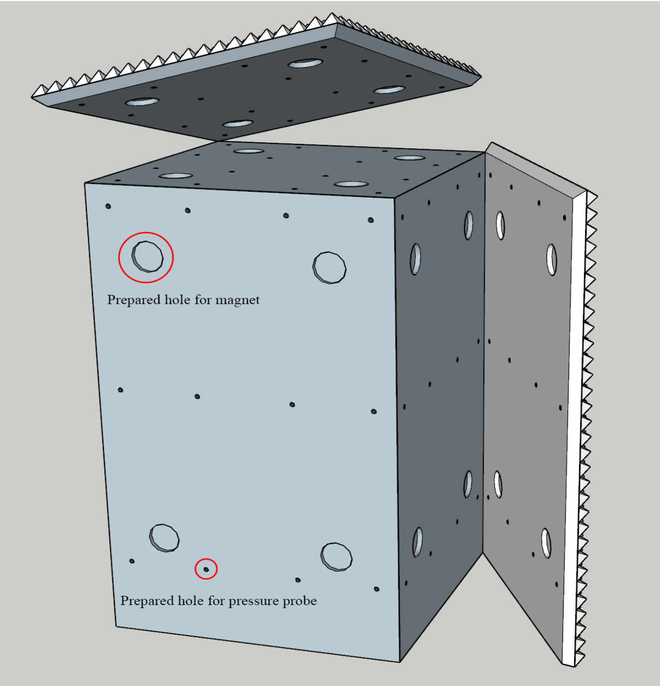

***Fig.** 7 Modular model with 4 mm geometric wall roughness.*

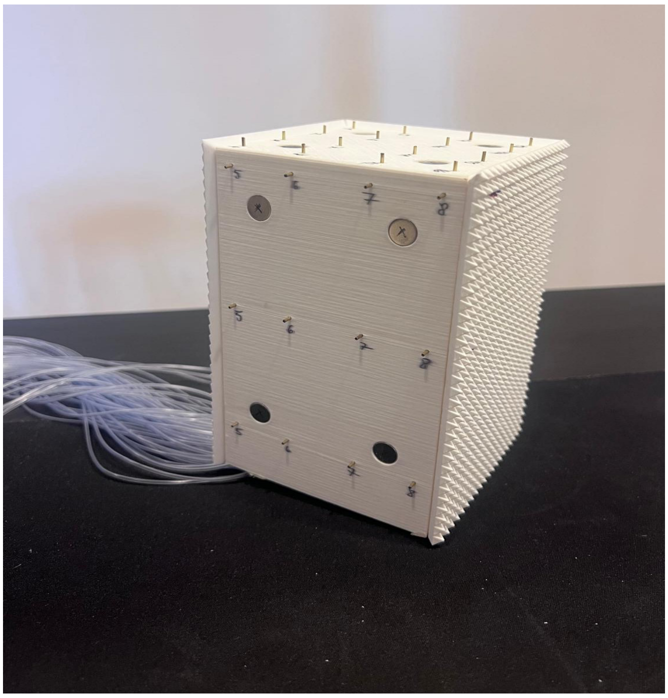

***Fig.** 8 Real modular model with 4 mm geometric wall roughness.*

for the precise and secure mounting of the interchangeable wall panels. Each of these panels was 3D printed separately and contained the same probe holes and magnetic connectors, thus ensuring fast and repeatable installations.

This modular system enabled the use of a single core body to which surfaces with following roughness heights were attached sequentially: 0 mm, 0.5 mm, 1 mm, 2 mm, 3 mm, and 4 mm. This approach significantly simplified the experimental preparation, reduced manufacturing time, and ensured consistency between the test configurations.

The model was mounted on a rotating table with an adjustable wind azimuth, which allowed for testing in various wind directions. For the purpose of this experiment, a single wind direction matching the simulation setup was used.

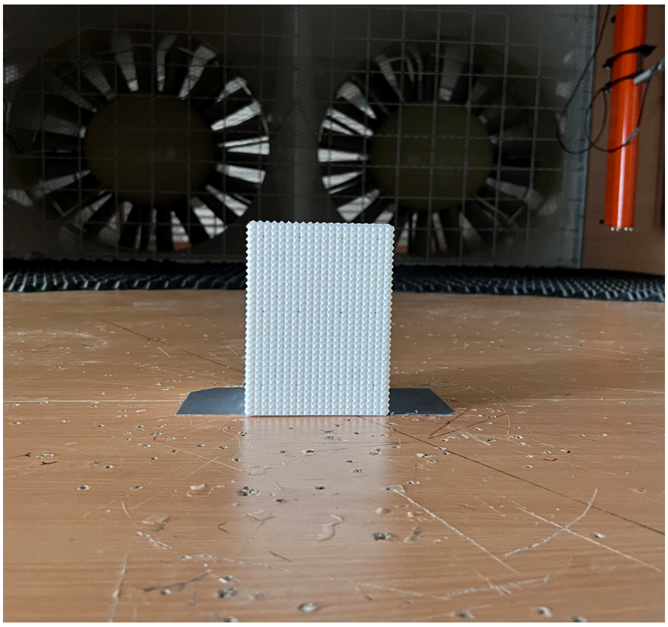

***Fig.** 9 Modular model installed in the wind tunnel.*

5.2 Measurement Points and Technique

The measurement points were systematically distributed across the entire surface of the model to capture the spatial distribution of the pressure coefficient (c_{pe}) at different heights.

Each of the four walls was equipped with 1 mm diameter probe ports. The measurement points were divided into four height levels:

- A1–A16: 10 mm above the base (lower level),
- B1–B16: middle of the model (75 mm from base),
- C1–C16: 10 mm below the top edge,
- S1–S16: on the top horizontal surface (roof).

Each level contained 16 evenly distributed points around the perimeter, which provided full coverage of the surface. The measurements were taken sequentially by switching the pressure probe between the ports, so that steady differential pressure values were recorded.

The measurement system included a precisely calibrated pressure probe connected to a data acquisition system capable of real-time digitization. All the measurements were carried out under steady conditions with at least 3 repetitions per point to ensure the reliability of the data and reduce any random deviations.
A differential pressure probe (Scanivalve dsa 3217), which was capable of capturing fast pressure fluctuations, was used. The measurements were performed after the flow stabilization following each fan restart.

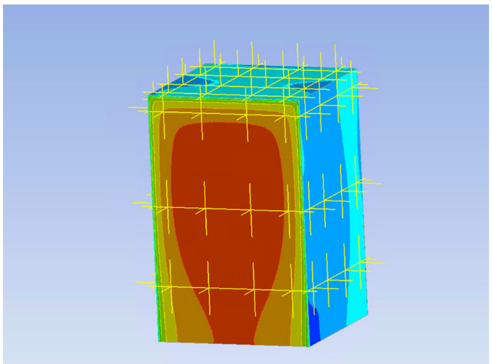

Fig. 10 Distribution of the measurement points on the model.

6 RESULTS

The results obtained include both numerical predictions and experimental measurements of the pressure coefficients (c_{pe}) across multiple roughness scenarios. The simulation data from Fluent, which used both geometric and constant roughness modeling approaches, were processed and compared with the wind tunnel measurements. The primary focus was on the accuracy of the c_{pe} values predicted and their agreement with the physical measurements under varying surface conditions. The data are presented in tabular and graphic forms to illustrate the performance and deviations of each method. A detailed discussion follows in the next section.

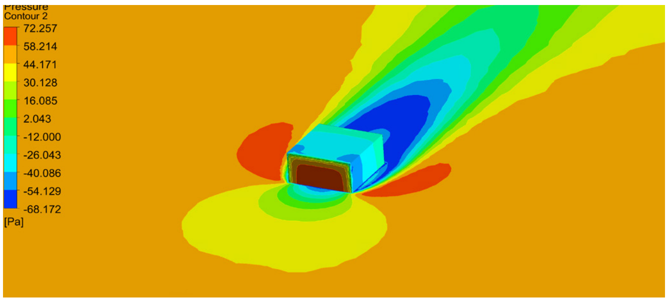

Fig. 11 Velocity contours in the domain around the building and pressure contours on the building itself.

6.1 Boxplot Comparison

Figure 13 shows a boxplot of the c_{pe} values across different roughness levels and modeling methods.

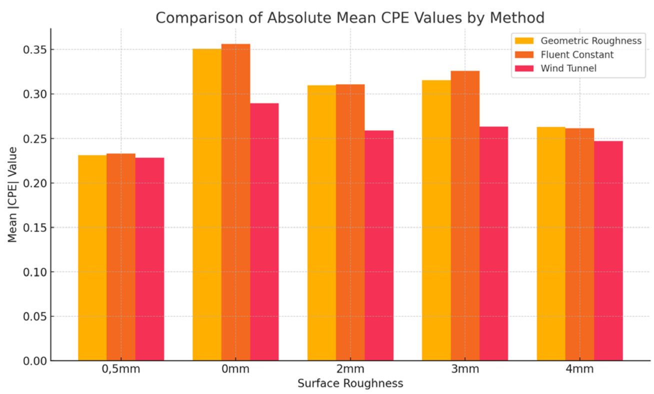

Fig. 12 Comparison of c_{pe} values by method and surface roughness.

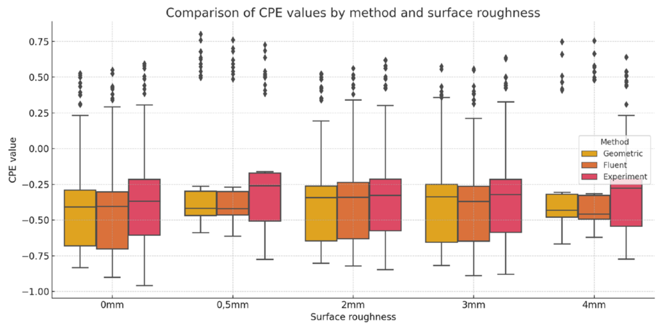

Fig. 13 Comparison of the c_{pe} values by the method and surface roughness. Dots represent individual c_{pe} values measured or computed at surface probe locations.

6.2 Mean Values

Tab.1 presents the average values of the pressure coefficient (c_{pe}) across all the surface roughness heights tested by comparing three approaches: the modeling of the geometric roughness, the fluent roughness constants, and wind tunnel measurements.

The values reveal a consistent trend where both numerical approaches tend to slightly underestimate the experimental results. This underprediction is most evident in the smoothest configuration (0 mm), where the geometric simulation yields a mean c_{pe} of 0.3508, and the Fluent constant simulation gives a value of -0.3559, both of which deviate more significantly from the wind tunnel measurement of -0.2898. This behavior is likely caused by the sensitivity of the development of the turbulent boundary layer in the absence of roughness elements, which is challenging to simulate accurately.

As the surface roughness increases, the differences between all three methods become smaller. For roughness values from 0.5 mm to 4 mm, both the geometric and Fluent-based simulations converge more closely with the experimental results. The Fluent model, in particular, demonstrates a good level consistency and tracks well with the physical measurements; this suggests that its empirical calibration is sufficient for approximating surface effects in these scenarios.

Overall, the trend across all the roughness configurations supports the conclusion that Fluent's simplified roughness representation can serve as a valid alternative to more computationally demanding geometric modeling, especially in engineering applications where absolute precision is less critical than a general level of accuracy.

Tab. 1 *Mean c_{pe} values by method and surface roughness*

Roughness	Geometric	Fluent	Experiment
0 mm	- 0.3508	- 0.3559	- 0.2898
0.5 mm	-0.2313	- 0.2332	- 0.2280
2 mm	-0.3100	- 0.3110	-0.2589
3 mm	-0.3154	- 0.3258	-0.2632
4 mm	-0.2629	- 0.2615	-0.2471

7 CONCLUSION

This study provides a comprehensive comparison of pressure coefficient (c_{pe}) values obtained through three distinct approaches, i.e., experimental measurements in a wind tunnel, CFD simulations with geometrically explicit roughness, and CFD simulations utilizing simplified surface roughness constants in ANSYS Fluent.

The results indicate that the use of Fluent's roughness constants yields c_{pe} predictions that are in good agreement with both geometrically modeled surfaces and wind tunnel data. Across all the surface roughness heights tested, the mean absolute deviations remained within acceptable engineering tolerances. Particularly noteworthy is the close correlation ($r = 0.954$) between the numerical and experimental data, which further validates the modeling approach.

However, the simulations revealed that the largest discrepancies occur in the 0 mm roughness case, which is a fully smooth surface. This is likely due to the increased sensitivity of the development of turbulent flow in the absence of geometric triggers, which can cause subtle shifts in boundary layer behavior that are not fully captured by empirical roughness constants alone.

Despite this, the differences between the Fluent-based and geometrically modeled results were minimal for all the other roughness heights. The numerical models not only replicated the experimental trends but also captured the spatial distribution of the c_{pe} values with a high degree of consistency.

From a practical perspective, the use of software-defined roughness significantly reduces the complexity and computational cost of CFD simulations. The geometric modeling of fine surface textures increases the density of the mesh and simulation time considerably, often with diminishing returns in its predictive accuracy. In contrast, properly calibrated roughness constants offer a streamlined alternative, which is sufficiently accurate for most engineering applications.

Based on these findings, we can conclude that replacing explicit geometric roughness with Fluent's roughness model is a viable and efficient alternative for simulating aerodynamic effects on buildings and other structures, especially when computational resources or time is limited.

REFERENCES

BLOCKEN, Bert, 2014. *50 years of Computational Wind Engineering: Past, present and future.* Journal of Wind Engineering and Industrial Aerodynamics, 129, 69–102.

CASTRO, I.P., 2003. *Rough-wall boundary layers: mean flow universality.* Journal of Fluid Mechanics, 439, 339–398.

HE, Jiaxun, et al., 2015. *Surface roughness effects on turbulent flow characteristics in atmospheric boundary layer.* Journal of Wind Engineering and Industrial Aerodynamics, 143, 1–14.

KRALIK, J., 2015. *3-D CFD Simulation of the Airflow over a Gable Roof with Different Elevations.*

LI, X., et al., 2016. *Field measurements and numerical simulations of wind pressures on buildings.* Building and Environment, 103, 13–26.

LI, Xian, ZHANG, Hui, and HUANG, Hong, 2018. *Evaluation of turbulence models in simulating wind flow around an isolated building.* Journal of Wind Engineering and Industrial Aerodynamics, 172, 120–130.

RAASCH, Siegfried, and SCHROTER, Michael, 2001. *PALM – A large-eddy simulation model for atmospheric and oceanic flows.* Theoretical and Computational Fluid Dynamics, 15, 213–232.

SMITH, J., 2023. *Roughness characteristics and aerodynamics.* Applied Sciences, 13(4), 1234.

TAMURA, Yukio, 2012. *Towards practical use of LES in wind engineering.* Journal of Wind Engineering and Industrial Aerodynamics, 104–106, 2–9.

TANG, Wenxuan, LI, Qi, and KAREEM, Ahsan, 2020. *Comparison of wind loads on a low-rise building obtained from full-scale, wind tunnel and CFD simulations.* Building and Environment, 170, 106616.

TOMINAGA, Y., et al., 2008. *AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings.* Journal of Wind Engineering and Industrial Aerodynamics, 96(10–11), 1749–1761.

WU, Yun, and STATHOPOULOS, Ted, 2019. *Wind tunnel tests and CFD simulations of wind pressures on a building with a rectangular plan.* Building and Environment, 148, 372–384.

YANG, Tao, and GU, Ming, 2016. *Influence of surface roughness on aerodynamic performance of building facades.* Journal of Fluids and Structures, 62, 350–366.

ZHANG, Lei, YAO, Jun, and LIU, Wei, 2017. *Numerical simulation of wind flow over complex terrain and comparison with wind tunnel measurements.* Atmospheric Research, 193, 182–196.

ZHOU, X., 2024. *Selecting roughness constants for ABL.* Journal of Wind Engineering, 234, 104812.