International Journal of Scientific Engineering and Science
Volume 2, Issue 6, pp. 31-34, 2018. ISSN (Online): 2456-7361

Comparing the AIJ Guideline to COST for Practical Application of CFD to Wind Environment around Single Buildings

Okafor Chinedum Vincent
Nnamdi Azikiwe University Awka, Nigeria

Abstract—*The significant developments in computer resources in recent past years has allowed computational fluid dynamics to serve as a very crucial method of predicting pedestrian wind environment for buildings within the atmospheric boundary layer during design stage in building construction. However, the accuracy of CFD simulation mainly depends on careful adherence to recommendation of best practice guidelines which provide valuable information on how CFD application should be used in order to reduce user errors accompanied by the incorrect use of CFD. This paper covers basic knowledge of CFD simulation for simple modeling case, such as flow around a tall building within the ABL using two empirical methods. From the comparison between the wind pressure distribution determined using the AIJ Guideline to the prediction from the COST guideline, the researcher found out that the performance of the two empirical methods were satisfactory.*

Keywords— *CFD; ABL; AIJ Guideline; COST Guideline.*

I. INTRODUCTION

Computational fluid dynamics simulation are being used by engineers for various wind engineering studies such as determining wind loads on buildings, evaluating wind flow patterns in built areas, predicting depression pattern in urban areas and evaluating pedestrian level wind comfort [1].

Recently, comprehensive literature reviews containing best practice guidelines on the use of CFD for wind around building phenomenon have been published in order to avoid or at least reduce user errors [2].

The most widely used recommendation proposed on the use of CFD in predicting the pedestrian wind environment includes the COST guideline by [3] and AIJ guidelines by [4]. The COST guidelines were mainly based on the results published by other authors summarized by [3] and [5]. The main purpose of COST Action 732 is the improvement and quality assurance of micro scale obstacle-accommodating meteorological models and their application to the prediction of flow and transport processes in urban or industrial environments [6].

The AIJ guideline proposed by the working group in the Architectural Institute of Japan were based on experiments from wind tunnel testing, field measurements and computations using different CFD codes to study the influence of various kinds of computational parameter for various flow fields.

According to [4], the difference between the AIJ guideline and COST guideline is that the former were derived from extensive cross-comparison while the COST guidelines mainly consist of results obtained from a literature review.

From the author’s point of view, wind engineering is not as much as developed compared to other branch of building technology in Nigeria.

Currently, there is no wind tunnel testing facility available in Nigeria to conduct wind analysis on buildings. The only available method of determining wind loading on high-rise buildings in Nigeria is through the use of wind loading codes which are in fact, adopted from other countries with some modifications to comply with prevailing conditions in Nigeria.

More so, there are very few published papers in Nigeria on this area with a critical review. Thus, the aim in this paper is to cover basic knowledge of CFD wind simulation around single buildings within the atmospheric boundary layer while also comparing the AIJ guideline proposed by [4] to COST guideline published by [5] for investigating pedestrian wind environment around single buildings within the ABL.

II. COMPUTATIONAL DOMAIN

A key aspect of modeling is the choice of the domain size and positioning of the single building within the domain [7].

Generally, the size of the entire computational domain depends on the targeted area and the boundary condition [8].

According to [3], for a single building, the distance from the top, to the computational domain should be 5H where H is the height of the Building. For the lateral boundary, 2.3H is required between the building sidewall and the computational domain leading to a maximum blockage ratio of 3%. [3] also recommends a distance of 5H to be used as maximum distance between the inflow boundary to the computational domain when the approach flow profile are well known and a distance of 15H for the outflow boundary between the leeward walls to the computational domain.

On the other hand, [4] suggested 5H for the lateral boundary and 10H for the outflow boundary, but their other boundary conditions recommended were similar to [3].

III. BOUNDARY CONDITIONS

When setting the inflow boundary conditions, the mean velocity profile and information about the turbulence quantities is needed to create an equilibrium boundary layer [3].

The boundary conditions recommended by [3] and [4] are as follows:

International Journal of Scientific Engineering and Science
Volume 2, Issue 6, pp. 31-34, 2018. ISSN (Online): 2456-7361

A. Inflow Boundary Condition

AIJ Guideline [4], recommends the use of a power law equation to represent the vertical velocity profile "$U_{(Z)}$" on flat terrain.

$$\boldsymbol{U_{(Z)}}=\boldsymbol{U_{(s)}}(\frac{Z}{Z_s})^{\alpha} \quad (1)$$

Where "$U_{(S)}$" is the velocity at reference height (Z_S). "α" is the power law exponent determined by terrain category; this exponent has a relation to roughness length as

$$\boldsymbol{\alpha} = [\mathbf{In}(\frac{Z}{Z_0})]^{-1} \quad (2)$$

The vertical profile of turbulent intensity is given by:

$$\boldsymbol{U_z} = \frac{\delta_{u(z)}}{U_{(z)}} = \mathbf{0.1}(\frac{Z}{Z_a})^{(-\alpha-0.05)} \quad (3)$$

Where "Z_a" is the boundary layer height determined by the terrain category and "δ_u" is the root mean square value of velocity fluctuation in stream-wise direction.

The turbulent kinetic energy and dissipation in the atmospheric boundary layer are given by:

$$\boldsymbol{K_{(z)}} = \frac{{\delta_{u(z)}}^2 + {\delta_{v(z)}}^2 + {\delta_{w(z)}}^2}{2} \cong {\boldsymbol{\delta_{u(z)}}}^2 = [\boldsymbol{I_{(Z)}} \boldsymbol{U_{(z)}}]^2 \quad (4)$$

$$\boldsymbol{\varepsilon_{(z)}} = C_u^{0.5} \boldsymbol{K_{(z)}} \frac{U_s}{Z_s} \boldsymbol{\alpha}(\frac{Z}{Z_s})^{(\alpha-1)} \quad (5)$$

Where C_u is the modal constant.

On the other hand, the COST guideline [3], recommended formulae suggested by [9] at the inflow boundary in which the vertical profile $U_z, K_{(z)}$ and $\varepsilon_{(z)}$ in the ABL assumes a constant shear stress with height as follows:

$$\boldsymbol{u_z} = \frac{u^*_{ABL}}{\kappa} \mathbf{In}(\frac{Z+Z_0}{Z}) \quad (6)$$

$$\boldsymbol{k_{(z)}} = \frac{{u^*_{ABL}}^2}{\sqrt{C_\mu}} \quad (7)$$

$$\boldsymbol{\varepsilon_{(z)}} = \frac{{u^*_{ABL}}^3}{\kappa(z+z_0)} \quad (8)$$

B. Lateral Boundary Condition

According to the AIJ guideline,[4] suggest an invicide wall condition to be applied at the lateral and top boundary as these boundaries do not affect the calculated results around the target building.

However, [3] recommended a constant shear stress at the top boundary corresponding to the inflow profile to prevent horizontal change from the inflow profile. Another option is to apply the values for velocity and turbulence quantities of the inflow over the entire top boundary as suggested by [10]. Symmetry boundary condition is only used at the top boundary if the domain top is outside the boundary layer.

C. Outlet Boundary

Both the AIJ guideline and COST guideline recommended an outflow boundary condition at the outlet boundary. This should be significantly far away from the building model so as to avoid errors accompanied with backflow.

D. Ground Boundary

At the ground, a logarithmic law with roughness parameter Z_0 or K_s can be used for the boundary condition

$$\frac{U_p}{U^*} = \frac{1}{K} \boldsymbol{In}(\frac{Z_p}{Z_0}) \quad (7)$$

Where $\boldsymbol{U^*} = (\frac{t_w}{\rho})^{0.5} = C_\mu^{0.25} \boldsymbol{K^{0.5}} \quad (8)$

In OPENFOAM CFD codes, the logarithmic law with roughness parameter K_s is expressed as follows:

$$\frac{u_p}{u^*} = \frac{1}{k} \boldsymbol{In}(\frac{\in z_p^+}{C_S K_S^+}) \quad (9)$$

Where "u_p" is the tangential component of velocity vector at a near wall node and C_S is a roughness constant which is set to ensure first matching order between the law of the wall and the inlet profile condition, ∈ is a smooth constant.

$$C_s = \frac{\in z_0}{K_S} \quad (10)$$

$$\boldsymbol{K_S}^+ = \boldsymbol{K_s} \frac{u^*}{V} \quad (11)$$

$$\boldsymbol{Z_p}^+ = \frac{u^* Z_p}{V} \quad (12)$$

For this analysis, $K_S = 20z_0$

IV. Test Case

The single Building used for the test case according to guidelines recommended by AIJ and COST respectively was a 48,768m (height) by 60.96(length) by 30.48m (width) Tall Building. The vertical profile used for the test case was determined from a reference wind speed of 3.40m/s at a reference height of 10m.

V. Solver Setting

SIM-FLOW commercial CFD code was used to perform the simulation. The 3D steady RANS equation was solved. The simple algorithm was used for pressure-velocity coupling, pressure interpolation was second order and second-order discritizaton scheme was used for both the convective terms and the viscous terms of the governing equation for fluid flow.

Steady state analysis used to develop the adaptive mesh was carried out using an RNG K-ε turbulence model.

VI. Result And Discussion

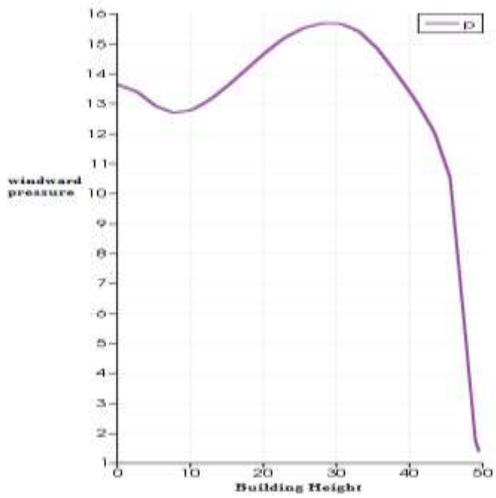

Fig. 1. Windward pressure according COST guideline.

International Journal of Scientific Engineering and Science
Volume 2, Issue 6, pp. 31-34, 2018. ISSN (Online): 2456-7361

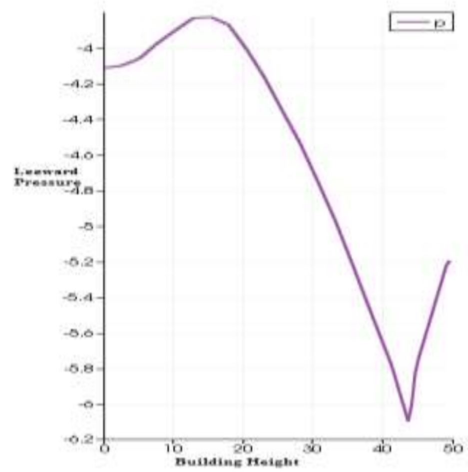

Fig. 2. Leeward pressure according to COST guideline.

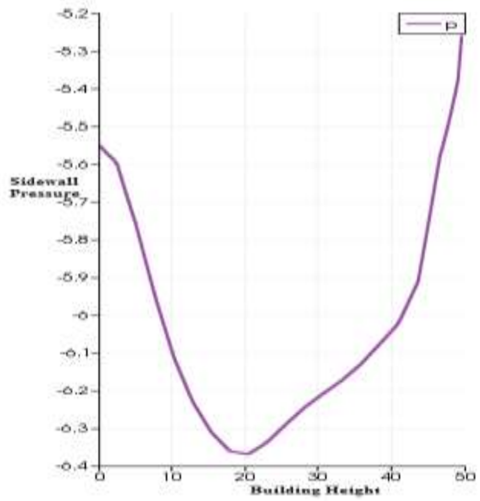

Fig. 3. Sidewall Pressure according to COST guideline.

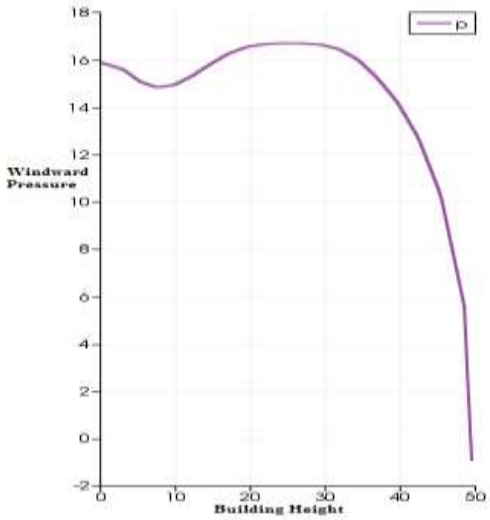

Fig. 4. Windward pressure according to AIJ guideline.

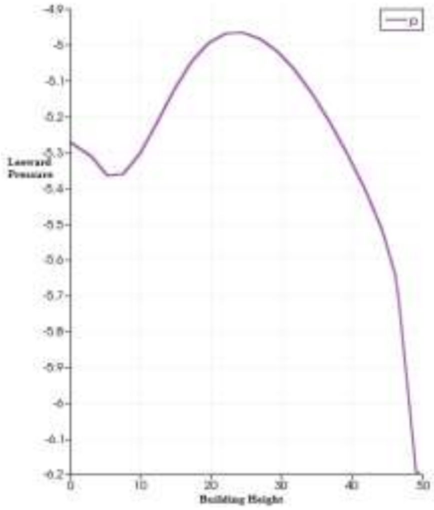

Fig. 5. Leeward pressure according to AIJ guideline.

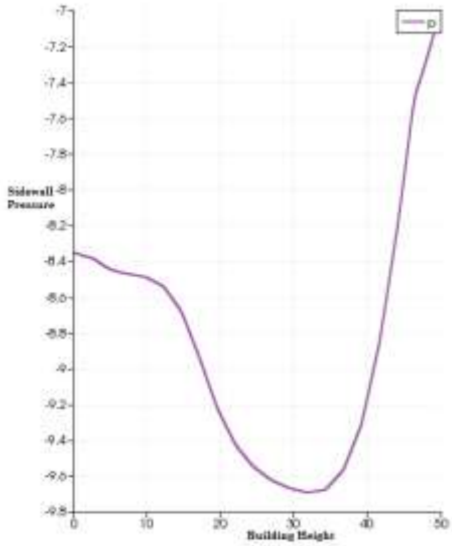

Fig. 6. Sidewall pressure according to AIJ guideline.

The CFD simulation was able to plot data of flow parameters for both analysis using the AIJ guideline and COST guideline respectively. Steady state incompressible flow solvers in OpenFOAM CFD code uses a kinematic pressure instead of a physical pressure in Pascal when solving the Navier stokes equation without explicitly showing the constant fluid density in the equation. As shown in the charts above, the pressure values displayed are kinematic pressures expressed in M^2/S^2 .Therefore, OpenFOAM users must multiply these values of kinematic pressures by the fluid density in order to calculate the pressure field expressed in Pascal(Pa).The maximum wind pressure recorded as per AIJ guideline for the windward, leeward and sidewall are 20.46pa,-6.084pa and -8.599pa respectively while pressure calculated using the COST guideline for the windward, leeward and sidewall are 19.58pa,-4.686pa and -6.442pa respectively.

VII. CONCLUSION

With the improvement in computer resources, computational fluid dynamics (CFD) simulation can be used as an effective tool in predicting wind environment around building during the preliminary design stage of a project. However, it is important to use this tool properly to obtain accurate results.

The proper control over main components of a CFD simulation, fluid domain, meshing and boundary condition as stipulated in the best practice guideline documents (AIJ and COST) would lead to an accurate simulation.

RNG K-ε model has some advantages over other turbulence model for this kind of analysis because of its superior responsiveness to the effect of streamline curvature, vortices and rotations. But tend to produce lower turbulence level and can underestimate the value of “K”. CFD simulation is usually able to capture flow visualization so as to be able to validate model results in wind simulation. However, it is recommended to use sophisticated Large Eddy Simulations (LES) if the situation demands more accurate detailed results.

REFERENCES

[1] A. U. Weerasuriya, “Computational fluid dynamics (CFD) simulation of flow around tall building,” *Engineer: Journal of the Institution of Engineers, Sri Lanka*, vol. 46, issue 3, pp. 43-54, 2013.

[2] C. V. Okafor, P. E. Ogunoh, J. U. Ezeokonkwo, and D. A. Obodo, “Atmospheric boundary layer simulation using wall function approach in openfoam CFD software,” *European Journal of Engineering and Science*, vol. 3, issue 2, pp. 1-6, 2018.

[3] J. Franke, C. Hirsch, A. Jensen, H. Krus, M. Schatzmann, P. Westbury, S. Miles, J. Wisse, and N. Wright, “Recommendations on the use of CFD in wind engineering,” COST ActionC14: Impact of Wind and storm on city life and Built Environment, von karman Instute for Fluid Dynamics, 2004.

[4] Y. Tominaga, A. Mochida, R. Yoshie, H. Kataoka, T. Nozu, M. Yoshikara, and T. Shirasawa, “Aij guidelines around buildings,” *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 96, issue 10-11, pp. 1749-1761, 2008.

[5] J. Franke, “Recommendations on the use of CFD in predicting pedestrian wind environment,” *In: the Fourth International Symposium of Computational Wind Engineering*, Yokohama, Japan, July 2006.

[6] M. Schatzmann and R. Britter (Eds). Quality assurance of microscale meterological models, COST 732 report, European Science Foundation, ISBN 3-00-018312-4, 2005.

[7] C. V. Okafor, “Application of computational fluid dynamics model in high-rise building wind analysis-A case study,” *Advances in Science and Technology and Engineering System Journal*, vol. 2, issue 4, pp. 197-203, 2017.

[8] D. Kim, “The application of CFD to building analysis and design: A combined approach of an immersive case study and wind tunnel testing,” PHD Thesis, Virginia polytechnic institute and state university. USA, 2014.

[9] P. J. Richards and R. P. Hosey, “Appropriate boundary conditions for computational wind engineering model using the K-e model,” *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 46-47, pp. 145-153, 1993.

[10] B. Blocken, T. Stathopoulos, and J. Carmeliet, “CFD simulation of the atmospheric boundary layer: Wall function problems,” *Atmospheric Environment*, vol. 41, issue 2, pp. 228-252, 2007.