CFD Simulation of Wind Load on tall Modified Building Shapes

Abdellah Idrissi
Laboratory of Energy Engineering and Materials (LEEM)
FST, Sultan Moulay Slimane University
Beni Mellal, Morocco
abdellah.idrissi001@gmail.com

Rachid El Amraoui
Laboratory of Energy Engineering and Materials (LEEM)
FST, Sultan Moulay Slimane University
Beni Mellal, Morocco
r_amraoui64@yahoo.fr

Hicham El Mghari
Laboratory of Energy Engineering and Materials (LEEM)
FST, Sultan Moulay Slimane University
Beni Mellal, Morocco
hichamelmghari1@yahoo.fr

Abstract— **The accurate evaluation and prediction of wind loads and proper mitigations are very important in reducing the adverse effects of wind in the built environment. For this purpose, the selected CFD simulation analyses the airflow around an isolated high-rise building. Adding, this study comprehensively investigates aerodynamic modification techniques applied to high-rise buildings by using adequate mathematical formulation and numerical approach. The velocity inlet condition was applied using user-defined functions (UDF). The findings prove a good agreement with their experimental counterparts' and illustrate the most data to make choice of the best cases (Basic, Corner Cut, Chamfered, and Tapered).**

Keywords—CFD simulation; High-rise building; Wind Load; Turbulence model

NOMENCLATURE

CFD: Computational Fluid Dynamics

CWE: Computational Wind Engineering

UDF: user-defined functions

SRANS: Steady Reynolds Averaged Navier Stokes

LES: Large Eddy Simulation

DES: Detached Eddy Simulation

U: mean flow velocity

u′: velocity fluctuations due to turbulence

μ: molecular viscosity

I. INTRODUCTION

Computational Fluid Dynamics (CFD) software is increasingly being used to predict the effects of wind on buildings and on the people in and around them. It is well suited to studying the effects of wind speed on pedestrian comfort within and around buildings. The technique is known as Computational Wind Engineering (CWE)[1].

The study explores the transition in wind engineering from traditional methods to computational fluid dynamics (CFD), leveraging advancements in hardware, software, and numerical modeling [2].

The study investigates turbulence models in computational fluid dynamics for predicting outdoor microclimate and thermal comfort in urban planning. It compares Steady Reynolds Averaged Navier-Stokes (SRANS) RNG k-ε, Large Eddy Simulation (LES), and Detached Eddy Simulation (DES) approaches in simulating wind flow around an isolated building [3] .

With advancements in construction, tall buildings adopt unconventional shapes, posing challenges from wind-induced excitations. The study explores aerodynamic modifications for tall buildings, categorizing them as minor (corner cut, rounding) and major (taper, set-back), aiming to mitigate vortex shedding phenomena and enhance structural stability in response to evolving architectural designs [4]

II. METHODOLOGY

This study the performances of the Steady Reynolds Averaged Navier Stokes (SRANS) RNG k-ε.is defined as the use of Computational Fluid Dynamics (CFD) analysis for wind engineering applications. the numerical simulation using ANSYS Fluent 19 R3. In the present study four corner modification of the building. The rounding, chamfer and corner cut dimensions were set to be 10 mm.

A. System description

The selected simulation case is the airflow around an isolated high-rise building. The detailed measurement of the flow field was performed in the wind tunnel under urban atmospheric boundary layer by Meng and Hibi (1998) [5] and used as one of validation benchmark experiments in the guidelines of the Architectural Institute of Japan (AIJ) [6], The computational domain size was 21b × 13.75b × 11.25b as shown in Fig. 2. The width and height of the domain were as the same size as the experimental wind tunnel setup [5]. The models were divided into two configurations: single modification representing a model with changes in plan area along the building height and

corner modification incorporating chamfer and corner cut along four corners of the building. The chamfer and corner cut dimensions were set to be 10 mm.
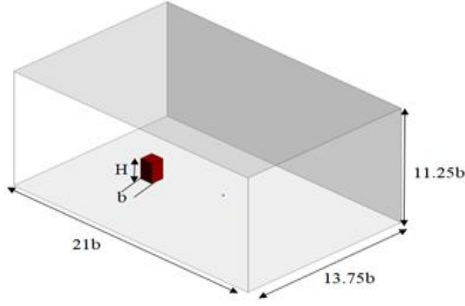

Figure 1. Computational domain

B. Maihematical formulation

To simulate the present model, several assumptions are considered, which are both the RANS approaches, the governing equations for the incompressible flow of a Newtonian fluid:

$$\frac{\partial \overline{u_j}}{\partial x_j} = 0 \qquad (1)$$

$$\frac{\partial (\overline{u_j u_j})}{\partial x_j} = -\frac{\partial \overline{p}}{\partial x_i} - \frac{1}{Re}\frac{\partial \overline{\tau_{ij}^{mol}}}{\partial x_j} - \frac{\partial \overline{\tau_{ij}^{turb}}}{\partial x_j} \qquad (2)$$

$$\tau_{ij}^{mol} = -2\vartheta \overline{S_{ij}} \qquad \overline{S_{ij}} = \frac{1}{2}(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}) \qquad (3)$$

$$u_i \frac{\partial \tilde{v}}{\partial x_i} = C_1 S_v \tilde{v} + \frac{1}{\sigma}\left(\nabla.[(v+\tilde{v})\nabla\tilde{v}] + C_2(\nabla\tilde{v})^2\right) - C_2 f_w [\frac{\tilde{v}}{d}] \qquad (4)$$

c. Modeling computational fluid dynamics
• *Mesh assessment*

Therefore, the computational domain was arranged in hexahedra grids with Three grids were generated for the SRANS simulations, including RNG-1, RNG-2 and RNG-3 it is essential to validate the correctness of the numerical model and the precision of numerical results. In a CFD analysis, the grid convergence study plays an important role in identifying the correct mesh The number of elements for three grids are 435405, 1963773, and 2552595.
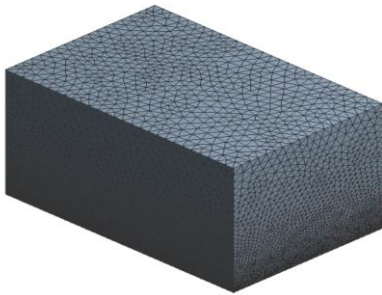

Figure 2. Mesh structures for CFD model

• *Numerical Simulation*

There are several boundary conditions considered to complete this numerical modeling and simulation. The inlet boundary is specified as a velocity inlet condition. This condition specifies the velocity of flow entering the domain. The velocity inlet condition was applied using user-defined functions (UDF) Here are some summaries of the problem definition and problem-solving steps in the table 1.

TABLEAU 1: OVERALL INPUT PARAMETERS FOR CFD SIMULATION IN ANSYS FLUENT 19.R3

Parameters	Inputs
Equation	Steady-RANS
Turbulence Model	RNG k-ε
Pressure velocity	SIMPLE
coupling	algorithm
Spatial discretization for pressure and momentum	Second-order
Inlet	(UDF)
Outlet	Pressure outlet
Initialization method	Hybrid

• *VALIDATION AND ANALYSIS*

Figure 3 shows the overall results from CFD simulation and exp by Meng and Hibi (1998) [5] and the validation model (CFD Validation) generated using ANSYS Since numerical results are affected by a range of factors, the result of the comparison of fig.3 shows the mean wind velocity distribution around the isolated building at the vertical plane according to the wind tunnel results (exp).
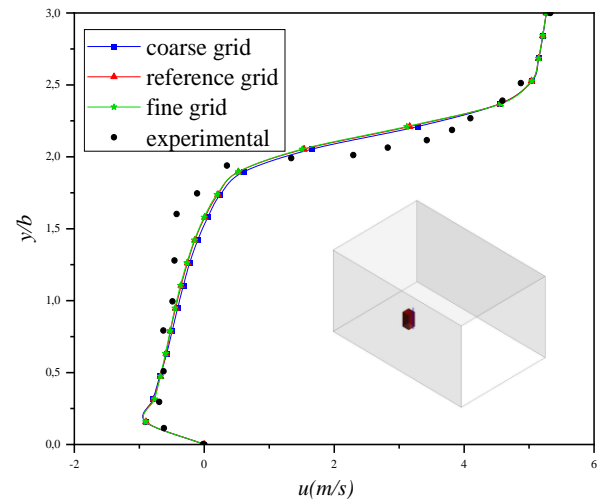

Figure 3. The mean wind velocity distribution (vertical plane)

around building

III. Results and discussion

Figures 4 and 5, depict the flow field structure around corner modification models, observed from both vertical and horizontal planes (specifically at 2/3 height). The alterations in the flow structure are directly influenced by the type and extent of the modifications. The visual analysis of Fig. 4 highlights that corner modifications induce three-dimensional changes in the flow structure, influencing the dimensions of the wake formed on the leeward side of the body. In comparison to the square model, there is a noticeable shift in the separation of the shear layer from the sides of the building.
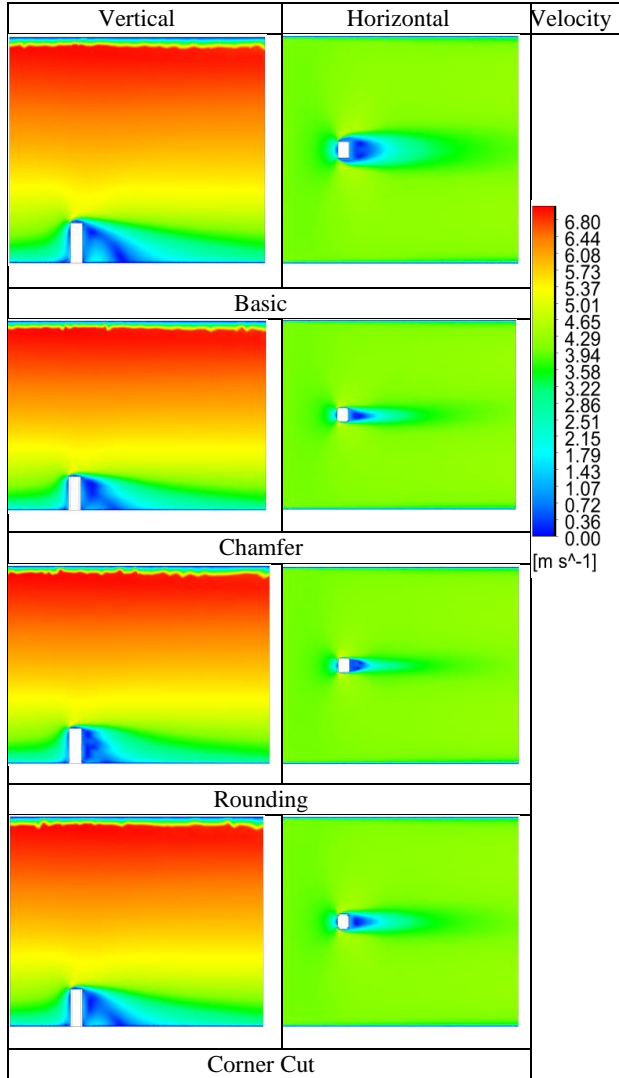

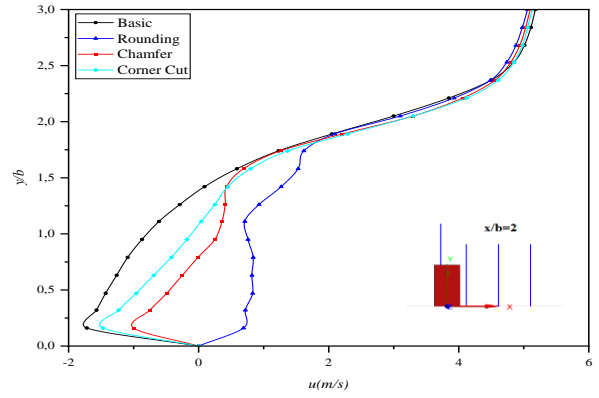Figure 4. Mean velocity contours on vertical and horizontal planes.

Figure 5. Mean velocity profile for x/b=2

The wake width of modified models is narrowed down which causes the increase of negative pressure (less negative) on the leeward side of the building model and consequently reduces the wind induced drag. The symmetry in shape contributes to a mean across-wind load of zero in CFD analysis, resulting in a substantial reduction in drag force.
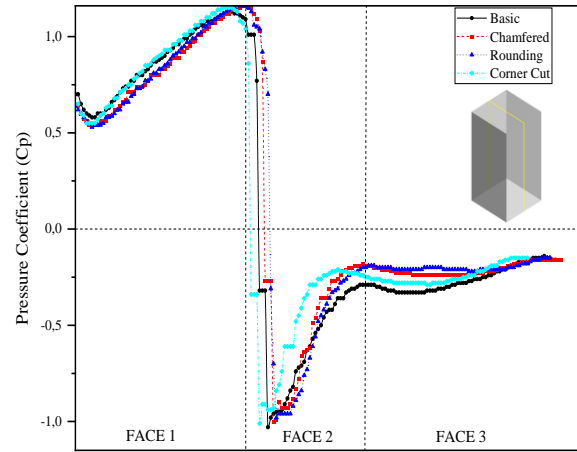

Figure 6. Cp profile for overall (corner modification)

Figure 6 shows the overall comparison distribution in terms of pressure coefficient for the basic, rounding, cut, and chamfer models. The rounding of the Cp distribution shows a similar trend to the basic models. However, a slight change in the Cp values occurs at every location for the three models.
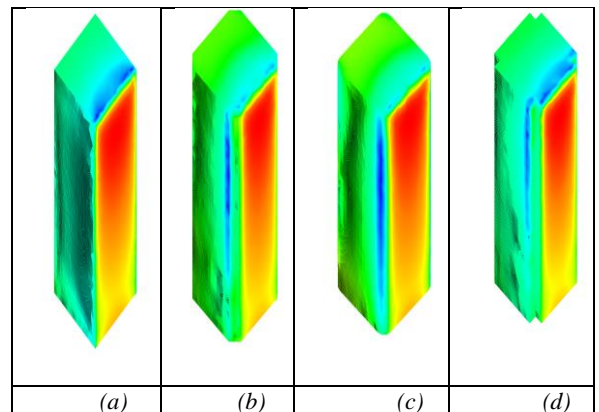

Figure 7: Pressure contour on the overall surfaces of the building models (a) Basic, (b) Corner Cut, (c) Chamfered, and (d) Tapered.

Figure 7 shows the pressure contour on the overall surfaces of the building models. All models develop -Cp at the side wall. The reduction of -Cp at the side walls of single corner modification can be associated with sharp corners on the

such, to break up the vortices and lose their coherence, softening to the sharp edges must be made by introducing corner cut or chamfer, Moreover, the rounding model performs better than the cut, and chamfer models in

basic model. In terms of suction, the introduction of corner cuts is more efficient than chamfering the corner. It is because buildings with sharp corners induced strong vortices or vortex shedding. As
reducing the suction at the side wall due to the reduction of the kinetic energy in Face 1 that was prolonged to the side walls and weakened the wake region.

IV. CONCLUSION

All models subjected to single modification performed better than the basic model in reducing the maximum +Cp and -Cp. In terms of suction, a corner cut was shown to be more efficient than a rounding. The cut, and chamfer models is more effective in reducing suction than geometry modification for high-rise structures can help mitigate aerodynamic concerns, particularly in pressure distribution on the building surfaces.

REFERENCES

[1] G. Palmer, B. Vazquez, G. Knapp, and N. Wright, "The practical application of CFD to wind engineering problems".
[2] "11ACWE-Dagnew".
[3] J. Liu and J. Niu, "CFD simulation of the wind environment around an isolated high-rise building: An evaluation of SRANS, LES and DES models," Build Environ, vol. 96, pp. 91–106, Feb. 2016. doi: 10.1016/j.buildenv.2015.11.007.
[4] A. Sharma, H. Mittal, and A. Gairola, "Mitigation of wind load on tall buildings through aerodynamic modifications: Review," Journal of Building Engineering, vol. 18. Elsevier Ltd, pp. 180–194, Jul. 01, 2018. doi: 10.1016/j.jobe.2018.03.005.
[5] Y. MENG and K. HIBI, "Turbulent measurments of the flow field around a high-rise building," Wind Engineers, JAWE, vol. 1998, no. 76, pp. 55–64, 1998.doi: 10.5359/jawe.1998.76_55.
[6] Y. Tominaga et al., "AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings," Journal of Wind Engineering and Industrial Aerodynamics, vol. 96, no. 10–11, pp. 1749–1761, Oct. 2008. doi: 10.1016/j.jweia.2008.02.058.