Validation of Computational Fluid Dynamics, Turbulent Flow Approach, for Wind Flow around Building

Dr. Salah R. Al Zaidee[1], Alfadhel B. Kasim[2]

[1]Instructor, College of Engineering, Baghdad University, Baghdad, Iraq

[2]M.Sc. student, College of Engineering, Baghdad University, Baghdad, Iraq

Abstract: *The wind loading is outcome of the pressure distribution on the structure in general. Pressure distributions due to wind are motivated by a wide range of aspects including urban surroundings, approach-flow conditions, and structure geometry. Computational Fluid Dynamics, CFD, is an important tool for calculating wind pressure coefficients, C_p, on structure cladding. This paper is established on a great number of CFD simulations to assess wind pressure coefficient. CFD simulations were carried out with Wind Load Simulator tool that found in Autodesk Robot Structural Analysis software. The study validation process is based on a comparison between CFD simulation results and experimental wind tunnel results that founded in the literature.*

Keywords: CFD, Wind Engineering, LES, CWE, ABL

1. Introduction

Wind is essentially caused by the heat gradient of the atmosphere because of changing in solar heating system of the earth's surface. It is initiated, by pressure rise between points of equal altitude or by density difference. Temperature and mechanical effect (shear effect) are resources of turbulence creation of atmospheric flow [1].

Computational Wind Engineering, CWE, primarily involves the purpose of CFD in wind engineering. CWE typically includes the combination of fluid problems like inflow turbulence, bluff body aero dynamics, grid generation, wake turbulence and high Reynolds number, Re, (between $10^7 - 10^9$). All these aspects need special consideration in numerical simulation. CFD is a numerical technique in which equations illustrating the fluid flow are resolved on a very high processer computer. Turbulent flow is distinct by the well known momentum, continuity, and energy equations, named after ***Navier and Stokes equations, NS equations,***[2].

Experimental and numerical methods are presented for studying wind flow phenomena, their performances in examining the flow are separated. Experimental tools like wind tunnel is classical method for exploring real flow phenomena by using a scale down model. Present knowledge of fluid dynamics has been primarily found from analytical and experimental feedbacks. The numerical method has progressed along with computer technology [3].

2. Review of Literature

The validation of CFD simulation has been studied by many researchers by various methods and the most important researches have been reviewed in below.

Dr S.K. Verma, et. Al. (2015) studied the numerical simulation for wind load on Octagonal Structures. Fluent-14 has been used in CFD analysis as tool for calculating wind pressure. Numerical analysis is carried out considering the same experimental study parameters that used. The outcome of this study is that the CFD offers to be an alternative to predict the wind phenomena on structures. Picturing of boundary layer separation can be detected precisely with the benefit of CFD simulation[4].

C. L. Fu, et. al. (2006) studied the validation of CFD simulation in estimating the wind loads on structure. CFD analysis was conducted to asses structural wind loads design. During the study, to validate the CFD results, experimental wind tunnel was made. This study shows that the CFD technique can be able to forecast the wind pressures of a single high rise building with a realistic accuracy [5].

A. K. Dagnew, et. al. (2009) presented a recent work on a computational assessment of wind pressures on tall structures. A more precise look has been occupied by this paper at the numerical calculation of wind pressures on tall structures by benefitting from the ***Common wealth Advisory Aeronautical Council, CAARC,*** structure model that exposes an appropriateness CFD tools for evaluation and detail description of complex structure aerodynamic features [6].

J. Šimicek, et. al. (2012) dealt with CFD analysis of wind load effect on buildings with theoretical basis turbulent and laminar flows in a boundary layer conditions and with the influences of wind during constructions. The numerical solution of wind flows has been executed with the Fluent software to obtain the velocity and pressure field at a steady wind speed in an inlet section. This approach can allow engineers to obtain a good picture of the pressure distribution around obstacles [7].

S. Ahmad, et. al. (2011) have carried out a 2-D numerical simulation for wind loads on structures. The model was prepared with FLUENT software in which full-scale model boundary layer, turbulence properties, and Reynolds number

have been simulated. This study has been illustrated that the numerical simulation holds a magnificent potential for extending codes of practice [8].

B. Parv, et. al. (2012) demonstrated that the CFD simulation presents similar results to those gotten from experimental wind tunnel. The focus of this study is to compare the wind pressure on a high-rise building using analytical model proposed by three design different codes and CFD simulation using ANSYS-CFX. The study found that CFD results is a very reliable, which can replace the experimental wind tunnel [9].

3. Theoretical Background of CFD Simulation

Computational fluid dynamics, CFD, is involved with numerical solution of the governing equations transport of energy, momentum, and mass, in moving fluids [10]. CFD involves substituting the partial differential equations, PDE, with algebraic equations which approximate the PDE. These equations arethen solved numerically to obtain values of flow field at the discrete points in time and space. While the *Navier–Stokes equations*, NS, are useableall over in the fluid flow field of the fluid continuum. The NS equations can be discretized then placed in algebraic form to solve by computer. The CFD simulation resolves for the related flow variables merely at the discrete points that build up the mesh or grid of the solution [11].

CFD activity developed and gained importance with accessibility of computers since 1960s. Nowadays, CFD finds widespread practice in applied research, and design of engineering structures. As Commercial software packages are wide spread, therefore, CFD become a significant tool of engineering practice [10].

A. Turbulent Flow Calculations
Movement of wind is turbulent. A short mathematical description of turbulence is hard to give, but it is enough to state that it happens in wind flow that has low viscosity about $1/6$ from water. Every movement of air at speeds more than $1\ m/s$ is turbulent, thus, initiating particles of air to transfer randomly in all directions [12].

Turbulence produces an appearance of vortices with inclusive range of time and length scales which act together in a dynamically complex mode. Methods of calculations of turbulent effect can be grouped into the following three categories:

1) Turbulence models for Reynolds-averaged Navier–Stokes (RANS) equations
This technique is focused on the mean flow and the effects of turbulence on mean flow properties. Before the application of numerical methods, the NS equations are time averaged. In the RANS, additional terms is included in flow equations to reflect the interactions among several turbulent fluctuations. These additional terms are modelled with traditional turbulence models. Between the best-known ones are the Reynolds stress and $k-\varepsilon$ model[13].

2) Large eddy simulation (LES)
This form of turbulence computations tracks the behavior of the larger eddies. The approach includes space filtering of the unsteady NS equations before the computations, which permits the larger eddies and discards the smaller ones. The effects on the determined flow (large eddies plus mean flow) because of the smallest unsolved eddies are involved by means of a supposed sub-grid scale model. Unsteady flow equations should be resolved because the demands on calculating resources in terms of volume and storage of computations are large. This method is starting to address CFD problems with complicated geometry [14].

3) Direct numerical simulation (DNS)
This technique calculates all turbulent velocity fluctuations and the mean flow. The method is not used for engineering flow computations because this calculation is extremely costly in terms of computing resources. Figure 1below shows the increase of computational cost for three types of turbulence calculations [15].
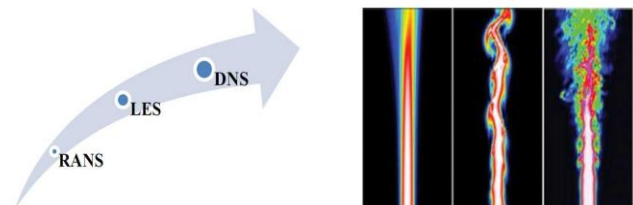

Figure 1: Increase of computational cost per type of turbulence modeling [16].

B. Atmosphere Boundary Layer (ABL)
A better gratitude of the structure of the ABL flow can be found by considering what happens to a fluid particle which flows into the ABL. As is mention in Figure 2, a rectangular particle holds its original form as it flows in the constant flow outside of the ABL. When it enters the ABL, the particle starts to distort due to the velocity gradient inside the boundary layer, the highest level of the particle has a greater speed than its lowest level. The particles do not rotate as they flow over the boundary layer. However, they begin to rotate when they pass over the fictitious boundary layer and enter the area of viscous flow. The flow is supposed to be irrotational outside the ABL and rotational inside the ABL[15].
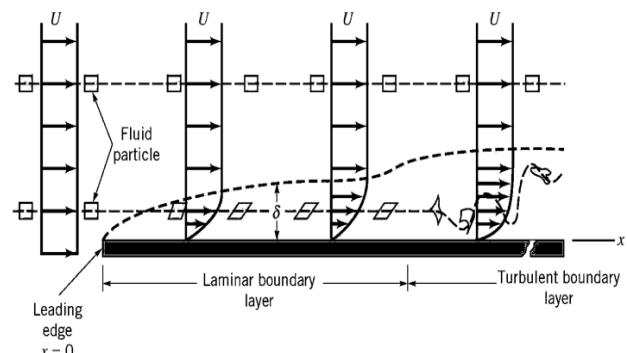

Figure 2: Distortion of a fluid particle as it flows within the boundary layer **[15]**.

Wind flows that have effect on structures are categorized by two essential features: the atmospheric turbulence and the

rise of the wind speeds with height as shown in Figure 3 below. Codes of Practice reflect the atmosphere boundary layer with exposure that simulate upstream roughness [17].
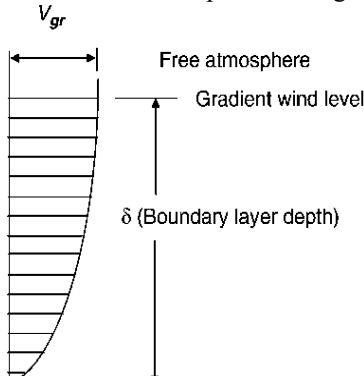

Figure 3: Schematic of the atmospheric boundary layer [17].

4. Validation Process

The wind load simulator adopted by Robot Structural Analysis software provides the engineers with a simulation tool to be used for wind flow around structure in 3D. This tool picturing surface pressure contours and output resultant wind loads for purpose of structural analysis. For comparison against the experimental wind tunnel data which founded in the literature, pressure data was obtained at specific coordinates to match the conditions of the pressure taps on the experimental physical model.

a) Experimental Results
The complications of wind flow offered by the geometries of structures and through the features of the topography and obstacles upstream confirm the need to carry out comprehensive studies of wind pressures using CFD simulation and experimental wind tunnel models [3]. Emil Simiu, et. al. (1996) shows in Figure 4 and Figure 5 below the distribution of mean pressure coefficients, Cp, on an exploded cubical shape building, the wind impact perpendicular to the building surface and is assumed to have a constant and boundary layer velocity field respectively.
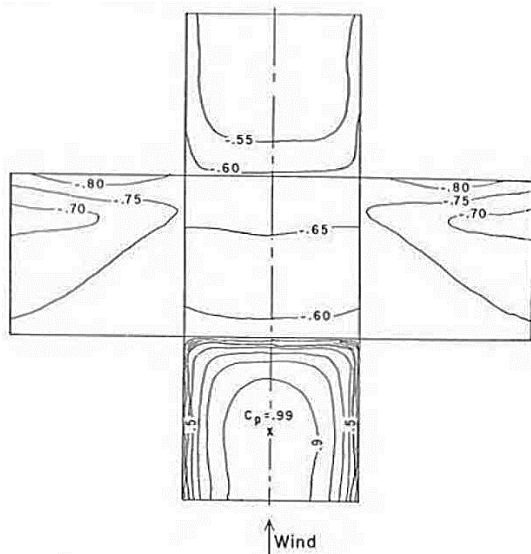

Figure 4: Pressure distribution on the faces of a cube in a constant velocity field [18].
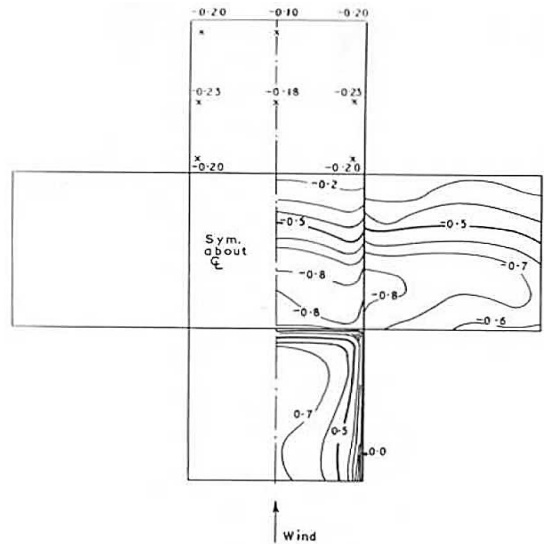

Figure 5: Pressure distribution on the faces of a cube in a boundary layer velocity field [18].

For tall buildings, Figure 6 and Figure 7 below shows mean pressure coefficients, obtained for constant and boundary layer velocity field respectively[18].
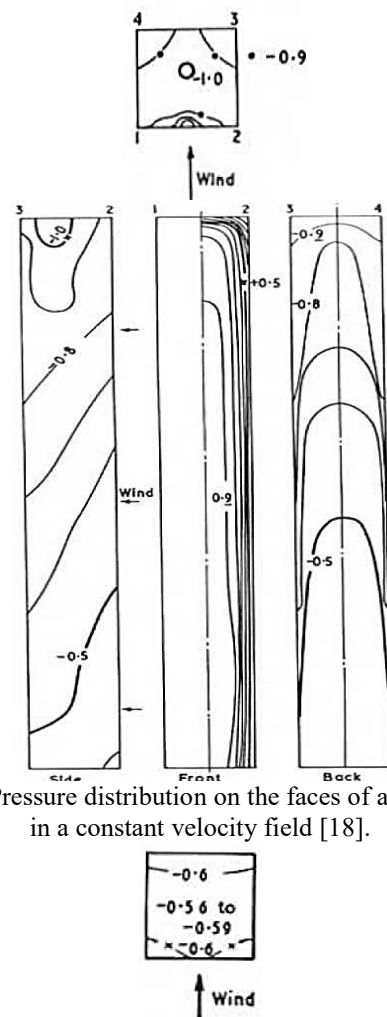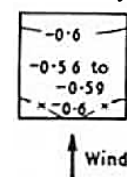

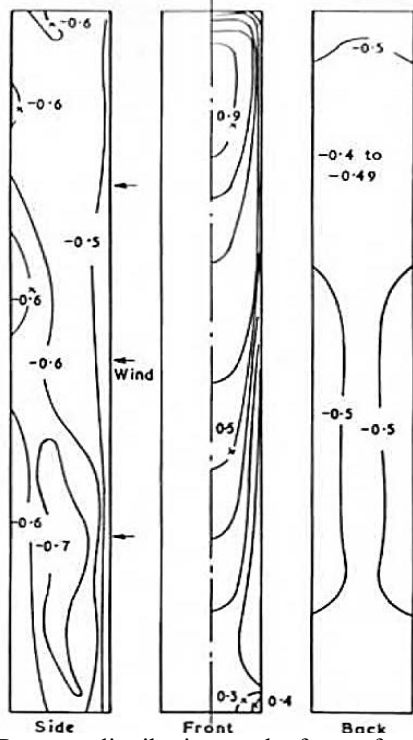**Figure 6:** Pressure distribution on the faces of a tall building in a constant velocity field [18].

Figure 7: Pressure distribution on the faces of a tall building in a constant velocity field [18]

b) CFD Simulation Results

During this study, a maximum wind of $40 \; m/s$ is adopted. Results of pressure distribution coefficient, C_p, determined with wind Load Simulator presented Figure 8 and Figure 9below for constant and boundary layer velocity field respectively. Dimensions of $30 \times 30 \times 30 \; m$are adopted for the cubical building. Exposure C from ASCE7-10 is adopted for Boundary layer velocity field [19]. For turbulence model, by default, wind load Simulator is use LES approach that discussed in section (A.2).
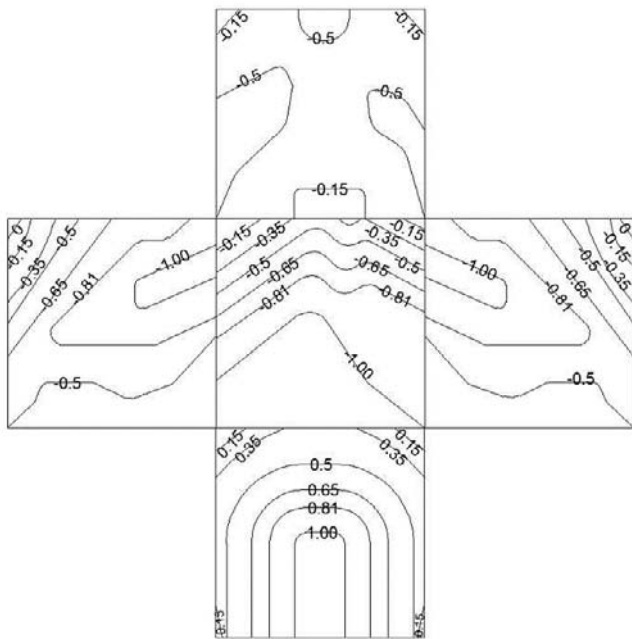

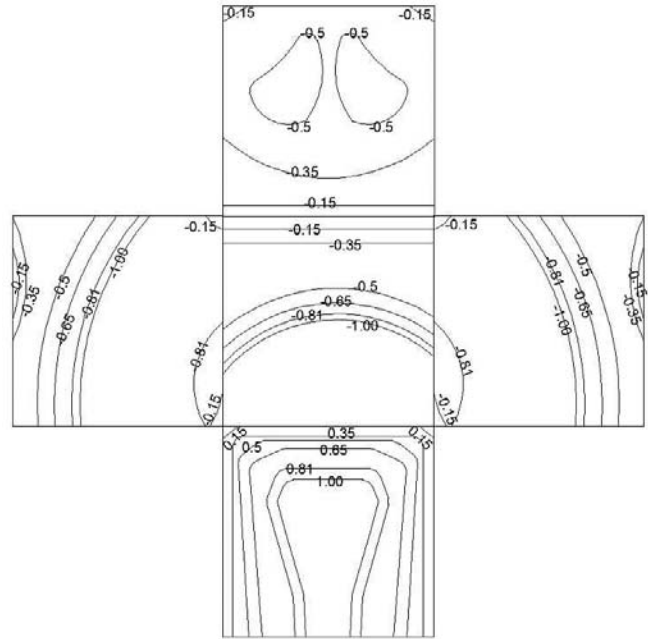**Figure 8:** Pressure contour on the faces of a cube in a constant velocity

Figure 9: Pressure contour on the faces of a cube in a boundary layer

For tall building, results of pressure distribution coefficients contours, C_p, are presented in Figure 10for constant velocity field and in Figure 11forboundary layer velocity field respectively. Dimensions of$30 \times 30 \times 90$ m is adopted for the tall building.
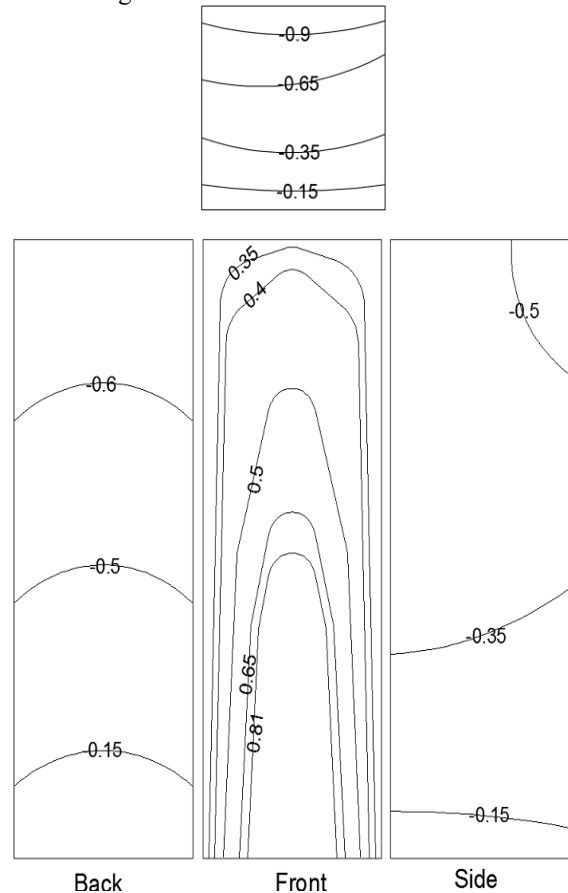

Figure 10: Pressure distribution over the sides and top of tall buildings; constant velocity field
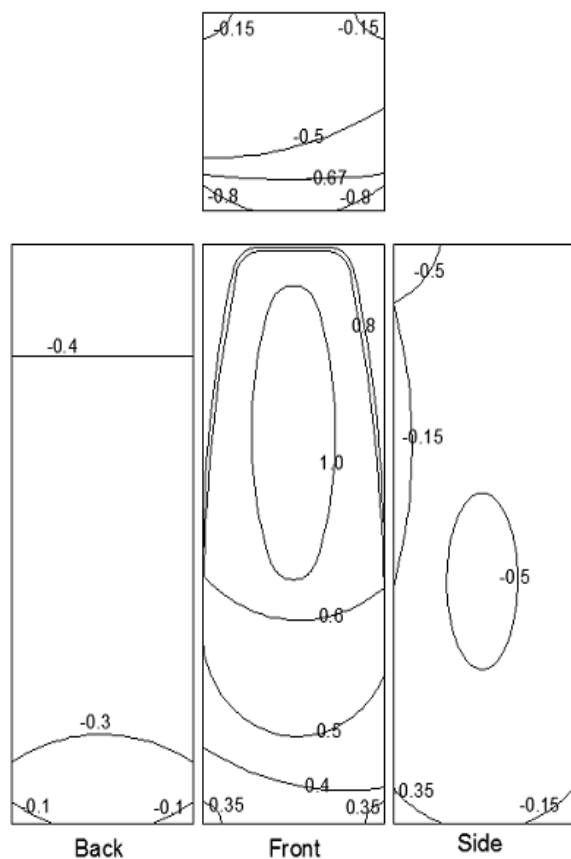
Figure 11: Pressure distribution over the sides and top of tall buildings; boundary layer velocity field

5. Conclusions

CFD simulation and experimental wind tunnel were compared out through case studies on a cube and tall building problem, to examine the suitability of using the CFD method to estimate wind pressure, Evaluation of wind pressures on a high-rise building shows that the CFD simulation can offer an accurate estimation for wind pressure on the front side. However, it tends to underestimate the wind suction on the backward side.

CFD offers to be an appropriate alternative to predict the wind pressure on structures and other various kind of buildings. Visualization of ABL separation can be detected precisely with the benefit of CFD simulation. The investigation of this study shows that the CFD can be used to forecast the mean wind pressure of a single building with reasonable accurateness.

References

[1] Claës Dyrbye and Svend Ole Hansen, Wind Loads on Structures, John Wiley & Sons, Ltd., 1997.
[2] Tamura, Yukio and Kareem, Ahsan, Advanced structural wind engineering, Springer Science & Business Media, 2013.
[3] T. Stathopoulos and C. Baniotopoulos, "Wind Effect on Buildings and Design of wind-sensitive structures," Springer Science & Business Media, 2007.
[4] Dr S.K. Verma, A.K Roy, Sunil Lather and Manoj Sood, CFD Simulation for Wind Load on Octagonal Tall, 2015.
[5] C. L. Fu, S. M. Lee and C. M. Cheng, "Validation of CFD simulations on the wind loads for tall buildings' preliminary design," in Proceedings of the 4th International Symposium on Computational Wind Engineering, 2006.
[6] A. K. Dagnew, G. T. Bitsuamalk and R. Merrick, "Computational evaluation of wind pressures on tall buildings," in Americas Conference on Wind Engineering, Puerto Rico, 2009.
[7] J. Šimicek and O. Hubova, " Nimerical Analysis of the Effect of Wind on Building Structures," Slovak Journal of Civil Engineering, pp. 28-34, 2012.
[8] S. Ahmad, M. Muzzammil and I. Zaheer, "Numerical prediction of wind loads on low buildings," International Journal of Engineering, Science and Technology, vol. 3, pp. 59-72, 2011.
[9] B. Parv, R. Hule and R. Zoicas, "Comparative Study of Wind Effects on Tall Buildings Using International Codes and CFD," in European Congress on Computational Methods in Applied Sciences and Engineering, Vienna, 2012.
[10] A. W. Date, Introduction to computational fluid dynamics, New York: Cambridge University Press, 2005.
[11] J. D. Anderson and J. Wendt, Computational fluid dynamics, Springer, 1995.
[12] B. S. Taranath, Structural analysis and design of tall buildings: steel and composite construction, CRC press, 2012, pp. 212-214.
[13] Wendt, John, Computational fluid dynamics: an introduction, Springer Science & Business Media, 2008.
[14] H. K. Versteeg and W. Malalasekera, An Introduction to Computational Fluid Dynamics: The Finite Volume Method, 2nd ed., Essex, England: Pearson Education Limited, 2007.
[15] B. R. Munson, T. H. Okiishi, W. W. Huebsch and A. P. Rothmayer, Fundamentals of Fluid Mechanincs, United States of America: John Wiley & Sons, Inc., 2013.
[16] B. Assaad, "Wind Effect on Super-Tall Buildings Using Computational Fluid Dynamics and Structural Dynamics," Florida Atlantic University, Florida, 2015.
[17] E. Simiu, design of buildings for wind a guide for ASCE 7-10, Johm Wiley & Sons, Inc., 2011.
[18] Emil Simiu and Scanlan, Robert H, Wind Effects on Structures: Fundamentals and Application to Design, New York: John Wiley & Sons, 1996.
[19] ASCE 7, Minimum Design Loads for Buildings and Other Structures, ASCE/SEI, 2010.