IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 15, Issue 2 Ver. IV (Mar. - Apr. 2018), PP 31-42
www.iosrjournals.org

Pressure Distribution Sensitivity To Turbulence Models For Cubic Buildings

W.Y. Abdelrahman[1],G.H. Mahmoud[2], M. M. El-Rakabawy[3],
1(Graduate student, Department of structural Eng. - Faculty of Eng. -Ain Shams University, Egypt).
2(Associate Professor of structural Eng. -Faculty of Eng. -Ain Shams University، Egypt).
3(Professor of structural Eng. - Faculty of Eng. –Ain Shams University, Egypt).
Corresponding Author:W.Y. Abdelrahman1

Abstract: *Numerical methods have been developed in the past decades to study the flow field and pressure distribution around buildings. The Computational Fluid Dynamics (CFD) technique is one of the most efficient methods. The CFD technique includes a wide range of turbulence models suitable for predicting airflow and mean pressure coefficient values. These models include Reynolds-Averaged Navier-Stokes turbulence models (RANS), Detached-Eddy Simulation (DES) model and Large-Eddy Simulation (LES) models. In this paper, a cubic building shape, which include five square faces, was studied to evaluate and validate the numerical results of various CFD turbulence models with available experimental works. The contours of wind pressure coefficients and wind flow around the cube have been determined using these different turbulence models. The results have been compared with experimental and other theoretical results. Further, the values of mean pressure coefficient (Cp) on cubic faces are compared with those values in codes and standard. The results are summarized and discussed.*

Keywords : *.Wind Pressure, Turbulence Models Sensitivity, CFD, Numerical Simulation, Cubic Building*

Date of Submission: 09-04-2018 Date of acceptance: 23-04-2018

I. Introduction

The use of atmospheric boundary layer wind tunnel to investigate the wind flow field around structures and to describe the distribution of wind parameters such as velocity, wind pressure etc. is the most accurate method. However, due to the complicity of the experimental wind tunnel, the elevated expenses and the limitations of obtained flow details due to the scaling effect maximize the need to use the numerical modeling techniques. The rapid advancement in computer hardware facilities and the development of software make the use of Computational Fluid Dynamics (CFD) technique efficient and powerful tool to describe a full-scale simulation of wind tunnel test with considerable achievement. [1-4]. By solving the conservation equations of mass, momentum, energy, and species concentrations, CFD can quantitatively calculate various air distribution parameters in an enclosed environment. It offers a higher level of flexibility, and lower cost than experimental studies.

There are many factors influencing the predicted numerical CFD results [5-10], among of them, the proper selection of turbulence modeling method, which is a key issue that will directly affect the simulation accuracy and efficiency. The choice of turbulence model will depend on some important considerations such as the physics encompassed in the flow, the established practice for a specific class of problem, the level of required accuracy, the available computational resources, and the amount of time available for the simulation. As observed from the former researches, the results are not always consistent. Contradicted results can be attributed to the differences in simulated cases, some other factors such as turbulence scheme, used grid, solvers efficiency, judging criteria, and human skills.

One of the most popular structural shapes studied and verified with field full-scale tests, small-scale boundary layer experimental results and numerical studies is the 6 m cubic shape. Shuzo [11] studied natural boundary layer flow over a cube using k-ε model considering the effect of boundary conditions and mesh sensitivity. He observed that the pressure distribution over the roof improved with the using of finer mesh on the windward face of the cube.

The RANS models, (k-ε, k-ω-SST) were applied by Köse and Dick [12] to predict pressure coefficient distribution around a 6 m cubic structure and verified their results with experimental data. Their results show that for both the windward face and leeward face, the RANS models present good agreement with previous results. Per contra, there were significant difference between the obtained results and experimental results for roof. In the same research, they used also, the DES model to predict flow over the studied cubic structure at low ($4x10^4$) and high ($4x10^6$) Reynolds numbers. At lower Reynolds number, the velocity field estimated accurately (no pressure data demonstrated). Nevertheless, for the higher Reynolds number, the DES turbulence model failed to predict the pressure coefficient on the roof.

For the higher value of Reynolds number ($4x10^6$), Haupt et al. [13] simulate atmospheric boundary layer flow over a cube using both the DES and zonal DES turbulence models. The used turbulence models captured very well the pressure coefficient on the windward face and leeward face. However, the results of the pressure coefficient of the roof is not accurate.

The application of LES model for bluff body flow performed by Kishan and Ferziger [14] to simulate fully developed channel flow around a cube to compare numerical results with experimental data provided by Martinuzzi et. al [15]. The efficiency of the used turbulent model (LES) proved through catching the mean velocity distribution contour, which has good agreement with the experimental result.

Many experimental work had been carried out to realize accurately the wind pressure distribution of modular cubic buildings. Among of these studies, the distinctive wind tunnel tests carried out on a small-scale model of the cubical Silsoe Experimental Building (SEB), [16]. The SEB was a (6 m x 6 m x 6 m) cube erected at Silsoe, in order to get full-scale data of the wind pressures acting on a building. This wind tunnel test was operated by Red Consultants Ltd, Hong Kong, working under the auspices of Department of Civil Engineering, Hong Kong Polytechnic University, Hong Kong. A scale of 1:30 was used to make the model of dimension 0.2m× 0.2m× 0.2m to determine the wind forces by conducting wind-tunnel experiments as shown in Figure 4. Pressure coefficient contours on the faces of Silsoe Experimental Building are illustrated in figure 5, for windward, leeward and top face

Richards et al [17] compared the CDF results for pressure distribution along the full scale 6 meters cubic structural surfaces with the wind tunnel tests. The results show good agreement for the coefficient of pressure for the case of wind perpendicular to one face for windward façade with wind tunnel tests. However, for the roof and leeward faces, there are some discrepancies in the values but the pressure distribution represented by contour lines is similar. They interpret the discrepancies to sensitivity to the scale, velocity profile, turbulence, variation in the cube to roughness and Reynolds number.

The aim of this research is to study and evaluate the different turbulence model performance and the sensitivity of the outcomes to each one of these models when applied to cubic building prototype. The target outcomes are the airflow and the mean pressure coefficient values. At the beginning, there is a brief introduction to the different techniques and turbulence models. Then the results of nine turbulent models in addition to the laminar flow as a basic control model, obtained through the application of CFD were compared with the available experimental and literature theoretical results on cubic shape prototype.

II. CFD Approaches And Main Types Of Turbulence Models

The CFD predict the turbulent flow field through three major approaches, namely Direct Numerical Simulation (DNS), which solve the highly reliable Navier-Stokes equation without approximations and consequently requires a very fine grid resolution to capture the smallest eddies in the turbulent flow. Large Eddy Simulation (LES), which separates the turbulent motion into large and small eddies such that this separation have no significant impact on the evolution of large-eddies. The large-eddies corresponding to the three-dimensional, time-dependent equations and turbulent transport approximations are made for small-eddies, which eliminates the need for a very fine spatial grid and minor time interval. Finally, the Reynolds-Averaged Navier-Stokes (RANS) approach which calculate statistically averaged (Reynolds-Averaged) variables for both steady-state and dynamic flows and simulates turbulence fluctuation effect on the mean airflow by using different turbulence models. There are different turbulent models generated from RANS approach such as the standard k-ε model due to its significantly limited requirements of hardware facility.

The governing equations of the flow are:

Mass conversation equation,

$$\frac{\partial ui}{\partial xi} = 0 \qquad (1)$$

Navier Stokes equation,

$$\frac{\partial ui}{\partial t} + u_j \frac{\partial ui}{\partial xj} = -\frac{\partial p}{\rho . \partial xi} + \frac{\partial^2 ui}{Re .(\partial xj)^2} + fi \qquad (2)$$

Where, *u*,*p*,*t*and R_e denote velocity, pressure, time and Reynolds number respectively and i,j refer to the three Cartesian coordinates system. General conservation (transport) equations for mass, momentum, energy, etc., discretized into algebraic equations. The discretized conservation equations solved iteratively. A number of iterations are usually required to reach a converged solution when the changes in solution variables from one iteration to the next are negligible, Residuals provide a mechanism to help monitor this trend, and overall property conservation achieved. The accuracy of a converged solution is dependent upon appropriateness and accuracy of the turbulence models, grid resolution and independence, and problem setup. ANSYS FLUENT 15.0 [18] software was used to perform steady flow computations based on a control volume approach for solving these flow equations.

With reference to the turbulence models, three main categories were developed. The first category is RANS models, which divided into eddy -viscosity models (EVM), and Reynolds stress models (RSM). The second category is LES models and third category is Detached Eddies simulation (DES) which combine both RANS and LES models and can substitute both of them. The description of used nine turbulence modes in this research summarized here:-
(a) RANS Turbulence Models
(i) RANS Eddy-Viscosity Models (EVM) which used two transport equations and include-
-Standard k- ε model
- Renormalization-group (RNG) k- ε model
- Realizable k- ε model
- Standard k- ω model
-Shear-stress transport (SST) k- ω model
(ii) RANS Reynolds Stress Model (RSM), which used seven transport equations.
(b) Large eddy simulation (LES) Turbulence Model
-Kinetic-energy transport model
(c) Detached eddy simulation (DES) model
- SST k- ω RANS model.
- Spalart-Allmaras model

III. CFD Numerical Simulation

1. Strategy of computational domain, mesh size and boundary conditions

The selection of computational domain size is a complicated task. The incorrect domain size, either bigger or small one may lead to wasting the computational time and resources or inaccurate results. In this research, the guidelines recommended and proposed by Blocken et al. [19], Franke et al. [20], and Sørensen and Nielsen [21] Irtaza1 et al. [22] are adopted. According to those studies, the minimum requirements for carrying out a consistent CFD simulation can be summarized in the following points:
• Second order schemes or above should be used for solving the algebraic equations.
• The scaled residuals should be in the range of 10^{-4} to 10^{-6}.
• Multi-block structured meshes are preferable and carrying out sensitivity analysis with three levels ofrefinements where the ratio of cells for two consecutive grids should be at least 3.4.
• Mesh cells to be equidistant while refining the mesh in areas of complex flow phenomena.
• Ifcells are stretched, a ratio not exceeding 1.3 between two consecutive cells should be maintained.
• Accuracy of the studied buildings should include details of dimension equal to or more than 1 m.
• The domain dimensions subjected to the following constraints, if H is the height of building, the domain lateral dimension equals to twice (H) in addition to building width, in the flow direction, the domain dimension equals to twenty time (H) plus the building length in that direction. A clear height above the building height (H) equals to three times (H), while maintaining a blockage ratio below 3 %.
• For the boundary conditions, the bottom would be a non-slip wall with standard wall functions, top and side would be symmetry, outflow would be pressure outlet and inflow would be a log law atmospheric boundary layer profile which should be maintained throughout the length of the domain when it is empty.

These simulation constrains are argued to yield consistent results when simulating wind flow round buildings. Accordingly, they are used in simulating wind flow around a surface mounted cube in turbulent channel flow to identify the validity of the simulation results in light of using these variables.

IV. Numerical CFD Setup

The preprocessor Gambit V2.4.6 was utilized to create a three-dimensional simulation for wind tunnel model. Figure (1-a), illustrates the flow direction while the geometrical properties of the three dimensional computational domain for cubic building model of characteristic side length of 6 miters demonstrated in Figure (1-b).

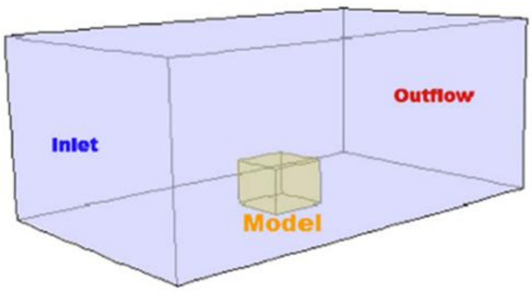


Fig.(1 –a) Schematic illustration of 3D Domain of Cubic building

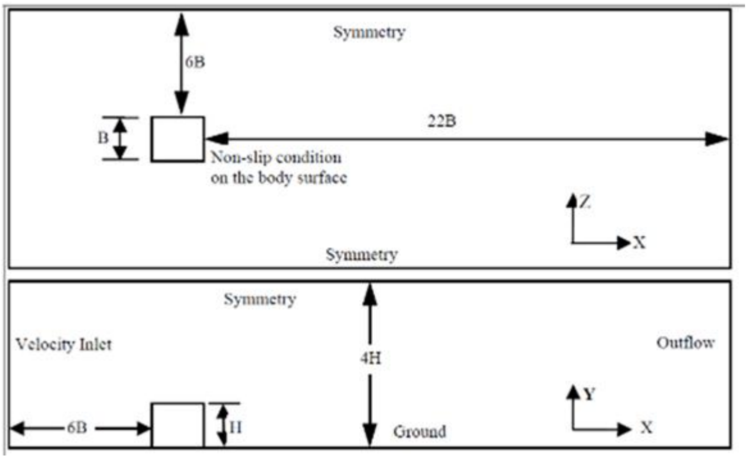
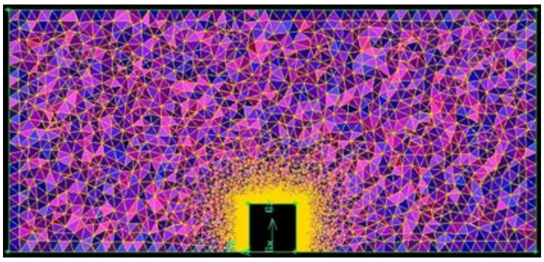
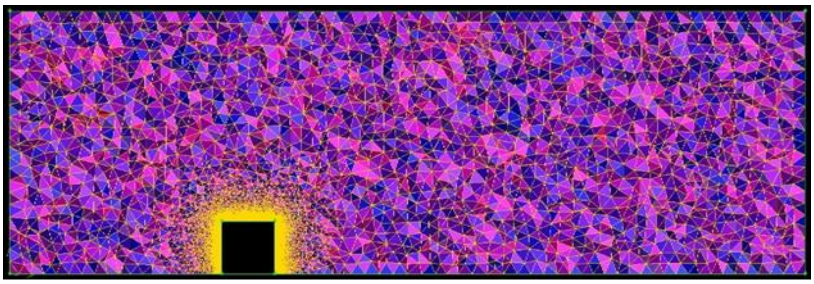


Fig.(1 –b) Dimensional properties of 3D model of Cubic building

A 3D tetrahedral mesh generated for the domain with 1,688,286 cells, 3,402,185 faces and 294,247 nodes and the domain boundary conditions consist of five walls for the cube building and four walls for the tunnel (top, bottom, right and left). Figure 2 shows two cross-section in the generated mesh for this model. The applied wind speed is 40 m/s, the fluid density is 1.225 kg/m^3 and the viscosity is 1.7894e^{-05} kg/m-s.



(2-a): Transversal Cross section at the middle of the cube



(2-b): Longitudinal Cross section at the middle of the cube

Fig.(2) Tetrahedral mesh arrangement

2. Studied Turbulences Models.

The most of the turbulence models incorporated in ANSYS FLUENT 15.0 [18] program have been used to predict wind flow properties around the previously described cubic building, also, the contour lines of pressure coefficient on the cubic faces were obtained. These models cover the basic laminar flow models as a control and the main three approaches of computational fluid dynamics (CFD) including Reynolds Averaged Navier-Stokes (RANS) modeling, hybrid RANS, Large Eddy Simulation (LES) and Detached Eddy Simulation, (DES).. The list of the used laminar and turbulence models are:-

(1) Spalart- Allmaras model.
(2) Renormalization Group (RNG) k- ε model.
(3) Realizable k- ε model.
(4) Standard k- ε model.
(5) Standard k- ω model.
(6) Shear-Stress Transport (SST) with k- ω model.
(7) Reynolds stress model (RSM).
(8) Detached Eddy Simulation (DES) with SST k- ω RANS model.

(9) Large eddy simulation (LES) with Kinetic-energy transport model.
(10) Laminar Flow model.

These ten turbulence models used to obtain the pressure distribution on the five cube faces and the results are compared with available experimental and literature theoretical results. Also a comparative study was made to verify the obtained results with the some wind loads codes

V. Results And Discussion

1. Wind flow around the cube and Contours of pressure coefficient (C_p) on cubic faces

The pressure coefficient $C_p$is defined by the following equation:-

$$Cp = \frac{(p - p_o)}{\frac{1}{2}\rho v^2} \quad (3)$$

Where$p$, p_o and v are the static pressure, reference pressure and the wind velocity respectively.

The results of the applied CFD turbulence previously mentioned ten models were obtained. Figure 3 illustrates longitudinal cross sections at the middle of cube showing the contours of wind flow pressure around the cube. In addition, the distribution of the contours of pressure coefficients on the windward and leeward cub faces are illustrated in Figures 4 and 5 for the ten computational fluid dynamics turbulence models respectively. Further, a comparison of the results of wind-tunnel experiments (Richards et al., [16]), and the applied ten CFD turbulence models can be seen in Figure 6.

The presented results in figures 3 to 6 are not perfect as might be hoped, but they reflect trends and some correct values in many cases. The distribution of the contours of pressure coefficients on windward face was acceptable, however, none of the turbulence models applied in this research could catch accurately the leeward and top face pressure distribution obtained through the wind tunnel experiments.

There are a few inconsistent experimental points both on the side wall and most noticeably on the roof of the cube that cannot be fully explained. These errors are most likely due to approach flow turbulence intensity variations. None of the models applied could accurately predict the experimentally obtained distribution on the leeward region. It appears that the under prediction of negative pressures are a consequence of over-prediction of the wake recirculation. No improvements were found with grid refinement. The above results confirm the need to accurately simulate the flow field around a bluff body, particularly the leeward wake region

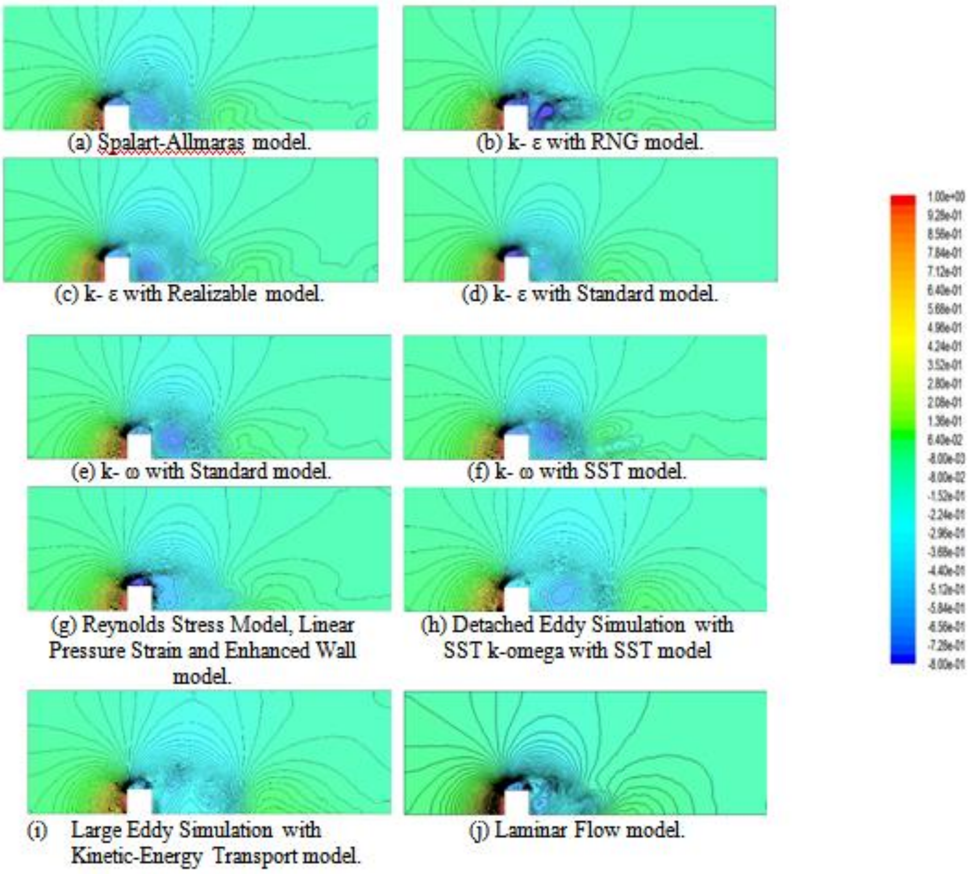
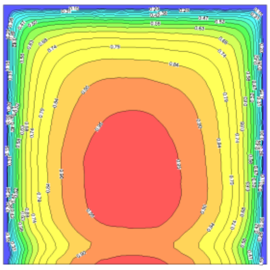
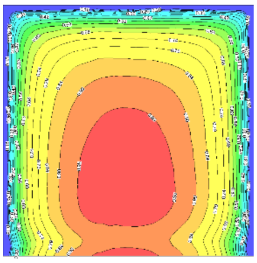


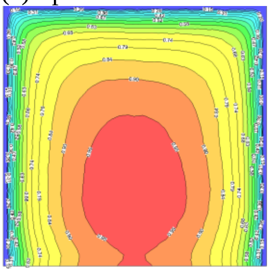
Fig. 3 Longitudinal cross sections at the middle of cube showing the contours of pressure coefficient



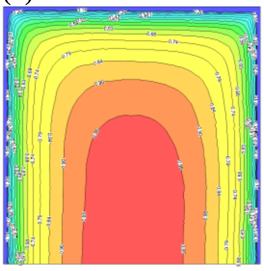
(a) Spalart-Allmaras model.



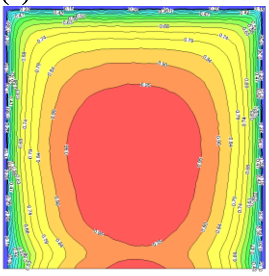
(b) k- ε with RNG model.



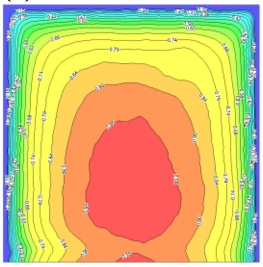
(c) k- ε with Realizable model



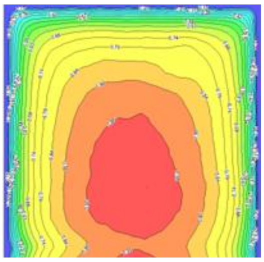
(d) k- ε with Standard model.



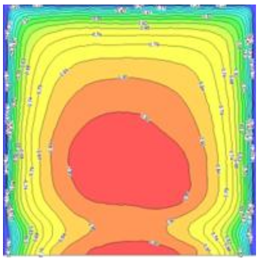
(e) k- ω with Standard model.



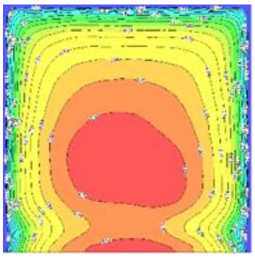
(f) k- ω with SST model.



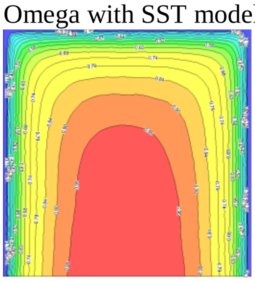
(g) Reynolds Stress Model, Linear Pressure Strain and Enhanced Wall model



(h) Detached Eddy Simulation with SST k-Omega with SST model

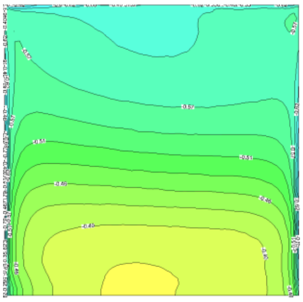


(i)Large Eddy Simulation with Kinetic-Energy Transport model

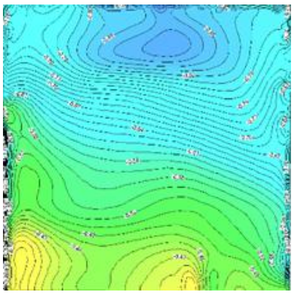


(j) Laminar flow model.

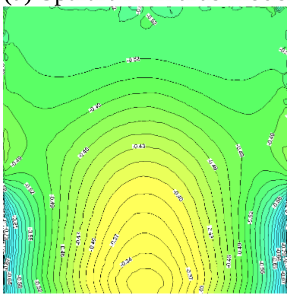
Fig. 4 Distribution of the contours of pressure coefficients on windward face



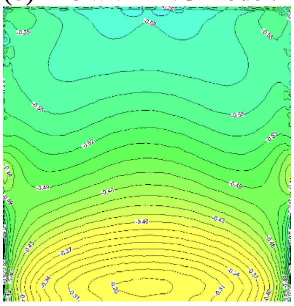
(a) Spalart-Allmaras model.



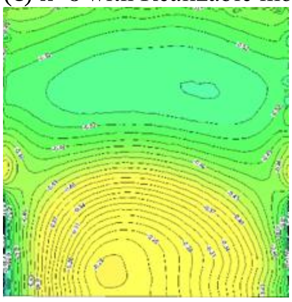
(b) k- ε with RNG model.



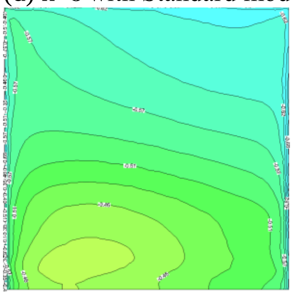
(c) k- ε with Realizable model



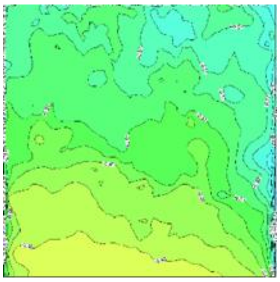
(d) k- ε with Standard model.



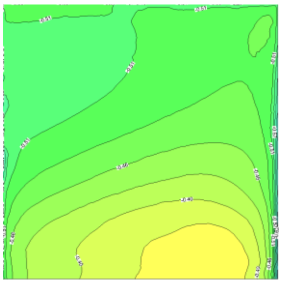
(e) k- ω with Standard model.



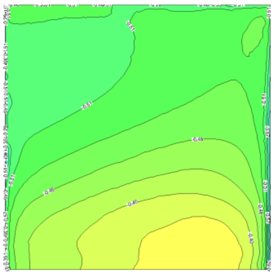
(f) k- ω with SST model.



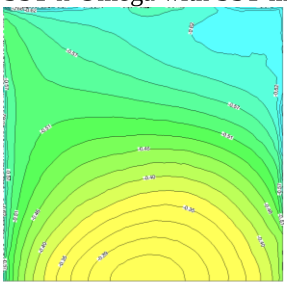
(g) Reynolds Stress Model, Linear Pressure Strain and Enhanced Wall model



(h) Detached Eddy Simulation with SST k-Omega with SST model

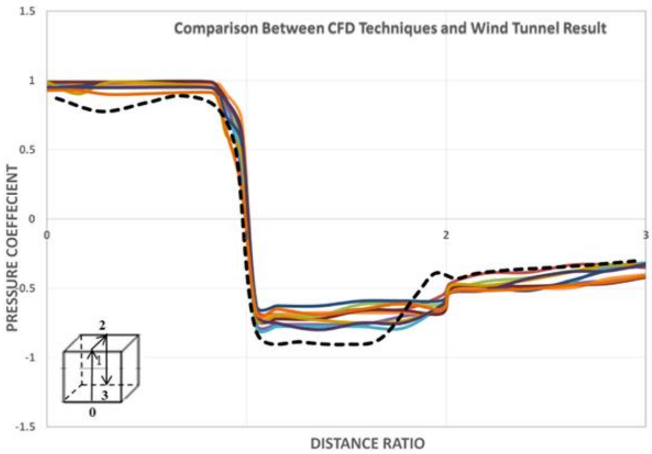


(i)Large Eddy Simulation with Kinetic-Energy Transport model

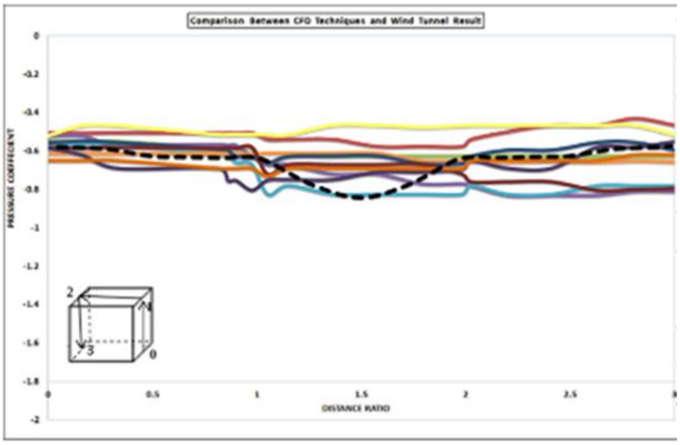


(j) Laminar flow model.

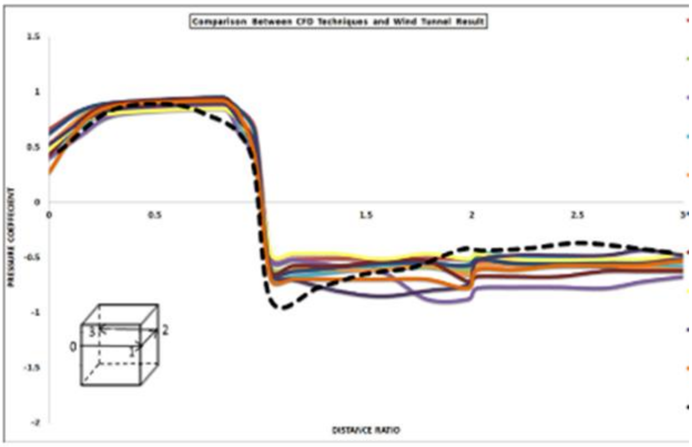
Fig. 5 Distribution of the contours of pressure coefficients on leeward face


Comparison Between CFD Techniques and Wind Tunnel Result
PRESSURE COEFFECIENT
DISTANCE RATIO

(a)


Comparison Between CFD Techniques and Wind Tunnel Result

(b)


Comparison Between CFD Techniques and Wind Tunnel Result

(c)

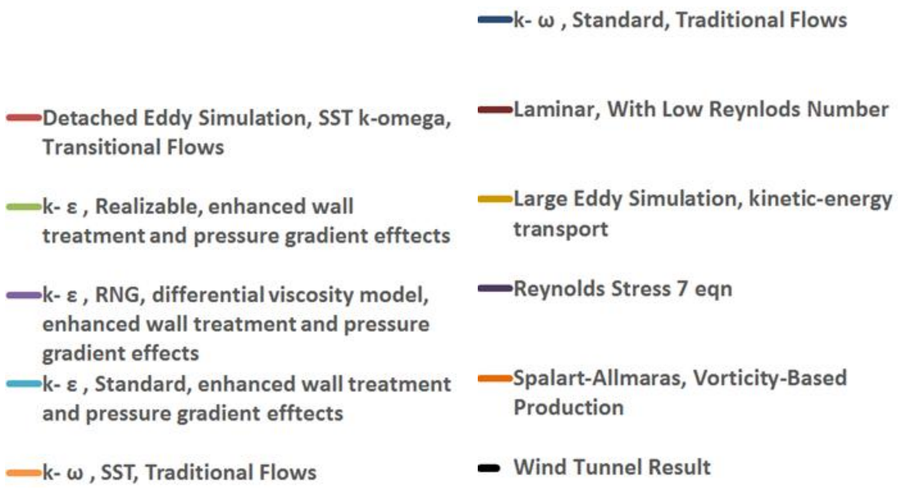


Fig. 6 Comparison between CFD Technique Results and experimental data for Predicting Pressure coefficient values on cube faces

2. Mean pressure coefficient (Cp) values and comparison with experimental results, Egyptian and ASCE codes values for cubic building faces

To facilitate the comparison between different codes and both experimental and different codes values, it is essential to get the mean values for the coefficient of pressure for each face, for each applied turbulence model. This step is very important as the experimental data is limited by the number of pressure taps used and the fact that codes allow one average value for each building face. The mean values of pressure distribution were obtained for each face (windward, leeward, top and side faces) for the different used turbulence models and summarized in table (1).

Table 1 Mean Pressure coefficient values on cube faces

	CFD Technique	**windward face**	**Leeward face**	**top face**	**side face**
1	Spalart-Allmarars	0.74	- 0.53	- 0.67	- 0.68
2	k- ε Model with RNG, differential viscosity	0.73	- 0.61	- 0.74	- 0.73
3	k- ε Model with Realizable, enhanced wall treatment and pressure gradient effects	0.75	- 0.49	- 0.63	- 0.62
4	k- ε Model with Standard, enhanced wall treatment and pressure gradient effects	0.76	- 0.46	- 0.79	- 0.70
5	k- ω Model with Standard, Traditional Flows	0.80	- 0.48	- 0.63	- 0.60
6	k- ω Model with SST, Traditional Flows	0.75	- 0.54	- 0.61	- 0.65
7	Reynolds Stress Model, Linear Pressure Strain and Enhanced Wall Treatment	0.74	- 0.46	- 0.70	- 0.65
8	Detached Eddy Simulation, SST k-omega, Transitional Flows	0.75	- 0.46	-0.62	- 0.62
9	Large Eddy Simulation, kinetic-energy transport	0.74	- 0.48	-0.66	-0.68
10	Laminar Flow Model	0.76	- 0.53	-0.67	- 0.69

The comparison between the mean pressure coefficient values on the cube faces using different CFD turbulence models have been compared with Egyptian [23] and ASCE7-10 [24] codes as demonstrated in Figures 7, 8, 9 and 10 for windward, leeward, top and side faces respectively.

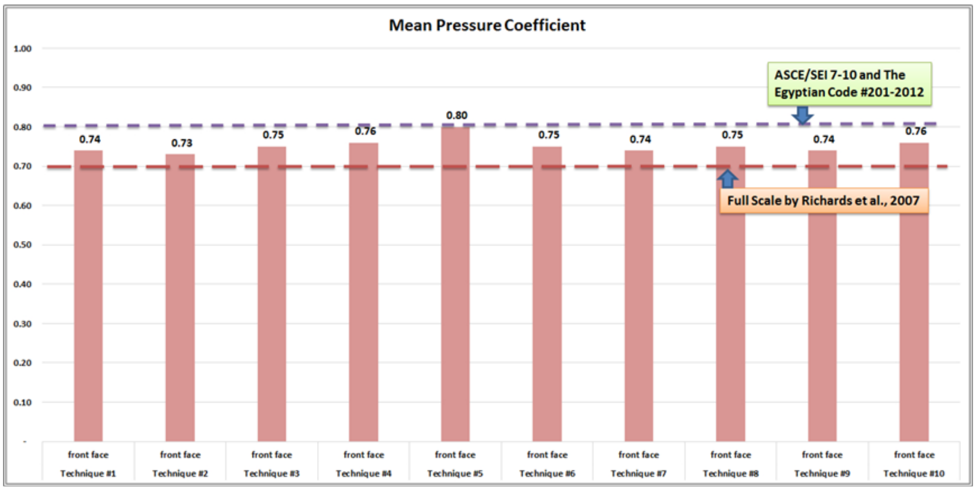


Fig. 7 Mean Pressure coefficient on the windward face

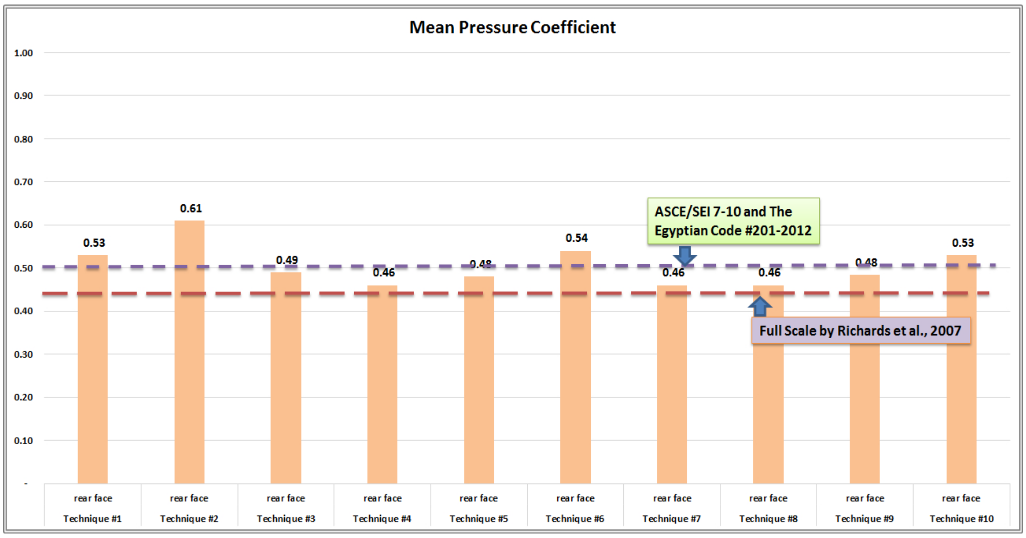


Fig. 8 Mean Pressure coefficient on the Leeward face, (-ve values)

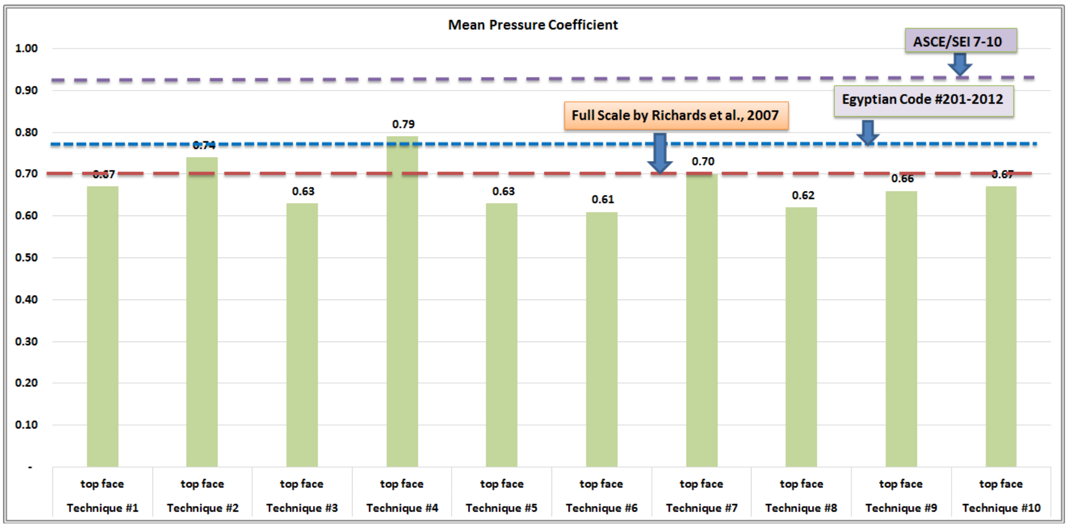


Fig. 9 Mean Pressure coefficient on the top face, (-ve values)

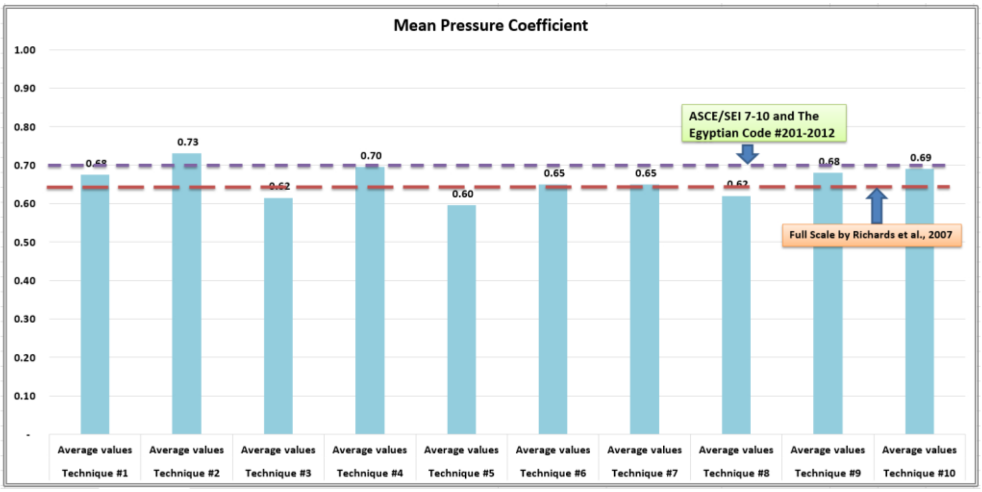


Fig. 10 Mean Pressure coefficient on the side faces, (-ve values)

It can be seen from the results in table 1 and figures 7 to 10 the following important notes:

- For both windward and leeward faces, the results of ten CFD techniques for the mean pressure coefficient values are slightly more than the corresponding values from experimental data, figures 7 and 8. The mean pressure coefficients were approximately 5% to 12% percent higher than the experimentally obtained values.
- For top face, the obtained results of k- ε Model with RNG, and k- ω Model with Standard model considerably overestimated the experimental mean pressure coefficient values, figure 9, with ratios ranges from 6% to 12%. The results of other models are slightly less than the experimental results.
- For side faces, the results of the studied models to the experimental mean pressure coefficient values have fluctuated ratios less than 12%, figure 10.
- The values of mean pressure coefficient (C_p) on cubic faces in Egyptian and ASCE7-10 codes are satisfactory compared with the experimental mean pressure coefficient values.

VI. Conclusion

A cubic building shape, which include five square faces, was studied to evaluate and validate the numerical results of various Computational Fluid Dynamics (CFD) techniques with available experimental works. These models include Reynolds-Averaged Navier-Stokes turbulence models (RANS), Detached-Eddy Simulation (DES) model and Large-Eddy Simulation (LES) models. The CFD results have been compared with experimental results. These results are not as good as might be hoped, although they do display the correct magnitude and trends in many cases. The results also confirm the need to accurately simulate the flow field around a bluff body, particularly the leeward wake region.

Further, the values of mean pressure coefficient (C_p) on cubic faces in Egyptian and ASCE7-10 codes are satisfactory compared with the experimental mean pressure coefficient values.

References

[1]. Zhang, Z., Zhai, J.Z., Zhang, W., Chen, Q, Evaluation of Various Turbulence Models in Predicting Airflow and Turbulence in Enclosed Environments by CFD: Part 1-summary of Prevalent Turbulence Models, HVAC&R Res,2007, 853-870.
[2]. Zhang, Z., Zhai, J.Z., Zhang, W., Chen, Q, Evaluation of Various Turbulence Models in Predicting Airflow and Turbulence in Enclosed Environments by CFD: Part 2-comparison with Experimental Data from Literature, HVAC&R Res, 2007,871-886.
[3]. Montazeri, H., Blocken, B, CFD Simulation of Wind-induced Pressure Coefficients on Buildings with and Without Balconies: Validation and Sensitivity Analysis, Build. Environ. 60, 2012, 137-149.
[4]. Tominaga, Y., Mochida, A., Murakami, S., Sawaki, S,Comparison of Various Revised K-ε Models and Les Applied to Flow Around a Highrise Building Model with 1:1:2 Shape Placed Within the Surface Boundary Layer, J. Wind. Eng. Ind. Aerod. 96, 2008.
[5]. Biao Lia, Jing Liua,b, Feifei Luoa, Xiaoxin Manc, Evaluation of CFD Simulation Using Various Turbulence Models for Wind Pressure on Buildings Based on Wind Tunnel Experiments, 9th International Symposium on Heating, Ventilation and Air Conditioning (ISHVAC) Procedia Engineering 121,2015, 2209 – 2216
[6]. Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T., Yoshikawa, M., Shirasawa, T, AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings, J. Wind. Eng. Ind. Aerod. 96, 2008, 1749-1761.
[7]. Tominaga, Y., Stathopoulos, T., Numerical Simulation of Dispersion Around an Isolated Cubic Building: Comparison of Various Types of K-ε Models, Atmos. Environ. 43, 2009.
[8]. FLUENT User's Manual Version 6.3, Fluent Inc., UK,2006.
[9]. Islam Abohela, Neveen Hamza, Steven Dudek, Validating CFD Simulation Results: Wind flow around a surface mounted cube in a turbulent channel flow, PLEA2012 - 28th Conference, Opportunities, Limits & Needs Towards an environmentally responsible architecture Lima, Perú 7-9 November 2012.

[10]. Bert Blocken, 50 years of Computational Wind Engineering: Past, present and future, J. Wind Eng. Ind. Aerodyn.129, 2014, 69–102.
[11]. M. Shuzo, 3-D numerical simulation of air flow around a cubic model by means of the k-ε model, J. Wind Engineering and Industrial Aerodynamics, 31(2), 1988,283-303.
[12]. D.A. Köse and E. Dick, Prediction of the pressure distribution on a cubical building with implicit LES. J. Wind Engineering and Industrial Aerodynamics, 98, 2010, 628-649.
[13]. S.E. Haupt, J.Z. Frank and L.J. Pelitier, Detached eddy simulation of atmospheric flow about a surface mounted cube at high Reynolds number. J. Fluids Engineering, 133, 2011, 1213-1222.
[14]. B.S. Kishan and J.H. Ferziger, A fluid mechanician's view of wind engineering: Large eddy simulation of flow past a cubic obstacle. J. Wind Engineering and Industrial Aerodynamics, 67&68, 1997,211-224.
[15]. R. Martinuzzi and C. Tropea, The flow around surface-mounted, prismatic obstacles placed in a fully developed channel flow. J. Fluids Engineering, 115, 1993, 85-92.
[16]. Richards, P. J., Hoxey, R. P. and Short, L. J, Wind pressures on a 6 m cube", Journal of Wind Engineering and Industrial Aerodynamics, 89(14-15), 2001, 1553-1564
[17]. Richards P.J., Hoxey R.P, Pressures on a cube building', Journal of Wind Engineering and Industrial Aerodynamics,102,2012. 72-86.
[18]. Ansys, Ansys fluent 15.0, Inc Northbrook, 2013, 49–53. doi:10.1016/0140-3664(87)90311-2.
[19]. Blocken, B., Stathopoulos, T., Carmeliet, J. and Hensen, J, Application of CFD in building performance simulation for the outdoor environment: an overview, Journal of building Performance Simulation, 4(2), 2011, 157-184.
[20]. Franke, J., Hellsten, A., Schlünzen, H. and Carissimo, B, Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment, Action 732, 2007, Brussels: Hamburg.
[21]. Sørensen, D. N. and Nielsen, P. V, Quality control of computational fluid dynamics in indoor environments, Indoor Air, 13(1), 2003, 2-17.
[22]. Irtaza1, H., Beale, R.G., Godley, M.H., Jameel, AComparison of wind pressure measurements on Silsoe experimental building from full-scale observation, wind-tunnel experiments and various CFD techniques, International Journal of Engineering, Science and Technology Vol. 5, No. 1, 2013, 28-41.
[23]. Egyptian Code to calculate the Loads and Forces in Construction and Buildings Works, ECP 201, 2012.
[24]. ASCE/SEI, ASCE/SEI 7-10, Minimum Design Loads for Buildings and Other Structures,2010,. doi:10.1061/9780784412916.

W.Y. Abdelrahman "Pressure Distribution Sensitivity To Turbulence Models For Cubic Buildings." IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) , vol. 15, no. 2, 2018, pp. 31-42.