IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

A Pressure Comparison of Tall Building of Baghdad University Tower and Baghdad Tower between Wind tunnel and CFD

W. S. KARRAR[1], A. M. SHAYMAA[2], AND M. JASSIM[3]

[1] Engineer, Department of Civil, College of Engineering, Mustansiriyah, Iraq

[2]Associate Professor Doctor, Department of Water Resource, College of Engineering, Mustansiriyah University, Iraq,

[3] Lecture Doctor, Department of Civil, College of Engineering, Mustansiriyah University, Iraq,

Abstract

Present paper is devoted to study wind pressure on two high-rise building models. Computational fluid dynamic (CFD) model was used in three-dimensional (3D). Utilizing finite volume method to predict the wind pressures on tall building using specific boundary conditions. RNG k-ε turbulence model was used in the program to consider the wind turbulence for two models. Current numerical approach was verified by comparison the wind pressure results for different velocity with experimental wind tunnel tests for two models. This valid CFD model was applied in the simulation of the wind analysis for two models, which is located in Baghdad City, Iraq. In addition, the design pressure of the two models was compared. It is connected that wind pressure results for wind tunnel test have a good agreement with the wind pressure results of CFD for different velocity for two models.

Keywords: tall buildings, finite volume method, computational fluid dynamic, RNG κ-ε turbulence model, wind pressure on building, wind tunnel test.

Introduction

Certain improvements in novel techniques of construction in twentieth century established structures which are considered to be fairly lightweight, low damping, as well as flexible, that might be exposing the structures to the impact of wind actions. Wind engineering can be considered as a field with the goal of majorly creating tools for better understanding regarding the fluid's action on structures with the origins, which might be marking out to 1960s. Improving the understanding with regard to this presented work encouraged the structural engineers for designing as well as ensuring the structure's performance with is the subject to wind's action to be in proper limits throughout the structure's lifetime in the structural safety as well as the serviceability criteria [8]. There have been many approaches to analyze tall buildings with regard to the wind load after utilizing building codes, wind tunnel or numerical analysis using the software. wind tunnel test is considered as significant tool which is providing the engineers with the capability for estimating the intensity as well as the nature regarding forces of wind which act on complex structures like tall buildings, this test has been particularly of high importance in the case of surrounding terrain, also the structure's shape will result in complex wind flows which have not been totally addressed through certain simplified codes [7]. CFD can be considered as latest approach for determining the wind loading on the structures. CFD applies numeric for determining fluid's flow in specific domain, such method is operating on the basis of Navier-Stokes equations, that is considered as derivation with regard to the indications of Newton and Euler of the next 3-conservation rules, conservation of momentum, conservation of energy, as well as the conservation of mass. The major benefit regarding such approach are that the simulation could be early utilized in building process, simulations could be repeated easily, simulations can be controlled easily , also flow field around building could be determined easily. The drawbacks regarding such approach are numerical techniques are regularly limited through computer capacity, the approaches are providing numerical errors, the approaches are requiring significant modelling that will be introducing errors and results were badly verified. [6] examined the impact of wind flow around 3D buildings through the use of CFD. Steady Navier-Stokes equation was solved through the use of k-ε turbulence model in the simulation. Power-law velocity profile has been utilized to describe the incident wind. There are 4 flows were specified in this work and simulation results have been put to comparison with full-scale wind tunnel data. On the basis of such comparison, it has been specified that there has been accordance with regard to outputs which are confirming the wind flow's validation via CFD. [9] suggested wind pressure on the flat roof regarding high-rise building. Reynolds average Navier stoke equation with standard k-ε turbulence model was utilized in the modeling. The wind blowing has been suggested to be in oblique and normal directions. The estimation regarding wind pressure for 3 distinctive distance from roof edges to first grid line's center. The results specified that the roof surface might be classified in to 2 subregions on the basis of computed pressure. Furthermore, it has been indicated that oblique direction that is related to the wind blowing impacted more than normal direction in the pressure calculations. [2] examined the building model of Commonwealth Advisory Aeronautical Council (CAARC) which has dimensions of (30mx45mx183m) in the wind analysis. Such work has been on the basis of numerical approach with focus on the equations of Reynolds Averaged Navier Stokes (RANS) as well as the turbulence model of Large Eddy Simulation (LES). The pressures of wind on tall building have been estimated and put to comparison with wind tunnel measurement with regard to buildings. Re-normalized group RNG k-ε turbulence model are providing more effective accordance regarding the boundary layer wind tunnel results in comparison to standard k-ε. The matching of turbulent model was indicated between LES numerical results as well as experimental results on the windward face. At the same time, the agreement deteriorated to some extent at sidewalls and enhanced at leeward wall. [5] provided case study which has been conducted on (208m) tall building with rectangular geometry. Comparison related to the results acquired from the simulations of CFD to predictions provided through Australian wind design standards (AS1170.2) has been provided. The work studied the wind tunnel test costs have been fairly high, also carrying out the wind tunnel test at preliminary design stages has been uneconomical because of the building's shape will be typically modified a few time throughout preliminary stages, also this might be added to the testing cost. Therefore, utilizing CFD for predicting the wind load on building, particularly at preliminary design stage has been of high importance. The work indicated that alternative approach wind tunnel tests has been CFD applied the behavior of wind for tall buildings, also the results indicating significant precision and possibly utilizing CFD in analyzing wind with regard to the high-rise buildings at final as well as initial steps of design in future. [4] presented the CFD simulation software for optimizing building efficiency through studying natural ventilation for both thermal comfort and human health in residential buildings. Besides introducing methodologies used in this regard and ways of validating results such as wind tunnel and in-situ measurements, selecting the appropriate turbulence model and presenting best practices in performing simulation process. Youth housing in New Damietta has been selected as a case study to evaluate natural ventilation inside three different prototypes depending on the local weather station wind data. The study case consists of three models of residential buildings. Each model contains residential units with an approximate area of (70 m2) and a height of five floors. A wind tunnel validated k-epsilon turbulence model has been used for this simulation. They concluded that the results of this research emphasizes the importance of using simulation software as a credible tool in achieving thermal comfort energy efficiency in residential buildings. [3] studied a wind pressure on (406m) tall slender structure with circular cross-section by using CFD which was complimented with experimental data obtained via wind tunnel testing. They were used RANS turbulence models to obtain the pressure distribution and flow behaviour of the structure. The comparison between the all methods by using pressure coefficient Cp. They were shown that RANS turbulence models were adequate to predict wind induced loads for buildings and the pressure variations on the windward, leeward and crosswind faces indicated an agreement with experimental data. The results were shown that the CFD simulations provide a reasonable estimate with less computational cost when compared with wind tunnel experimentation, and Overall k-ω provided good estimates compared with experimental data where in most cases the difference between both were less than 10% and at maximum cases (leeward face) less than 30%.

Governing Equations for Wind

The governing equations in fluid dynamics can be applied to wind flow. Liquid or wind flows in CFD codes are governed by partial differential equations, which are based on the conservation laws for mass, energy and momentum. The following expressions are applied for three dimensional, steady and incompressible flows with constant viscosity.

$$\frac{\partial \rho}{\partial t} + \left[\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z}\right] = 0$$ Continuity equation (1)

$$\rho\frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x$$ Momentum equation (2-x)

$$\rho\frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yx}}{\partial x} + \frac{\partial \tau_{zy}}{\partial z} + \rho f_y$$ Momentum equation (2-y)

$$\rho\frac{Dw}{Dt} = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{zz}}{\partial z} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zx}}{\partial x} + \rho f_z$$ Momentum equation (2-z)

$$\tau_{xy} = \tau_{yx} = \mu\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)$$

$$\tau_{xz} = \tau_{zx} = \mu\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)$$

$$\tau_{zy} = \tau_{yz} = \mu\left(\frac{\partial v}{\partial y} + \frac{\partial w}{\partial y}\right)$$

$$\tau_{xx} = -\frac{2}{3}\mu(\nabla . V) + 2\mu\frac{\partial u}{\partial x}$$

$$\tau_{yy} = -\frac{2}{3}\mu(\nabla . V) + 2\mu\frac{\partial v}{\partial y}$$

$$\tau_{zz} = -\frac{2}{3}\mu(\nabla . V) + 2\mu\frac{\partial w}{\partial z}$$

$$\rho\frac{D}{Dt}\left(e + \frac{V^2}{2}\right) = \rho q + \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right) - \frac{\partial(up)}{\partial x}$$

$$-\frac{\partial(vp)}{\partial y} - \frac{\partial wp}{\partial z} + \frac{\partial(u\tau_{xx})}{\partial x} + \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(u\tau_{zx})}{\partial z} + \frac{\partial(v\tau_{xy})}{\partial x} + \frac{\partial(v\tau_{yy})}{\partial y} + \frac{\partial(v\tau_{zy})}{\partial z} + \frac{\partial(w\tau_{xz})}{\partial x} + \frac{\partial(w\tau_{yz})}{\partial y} + \frac{\partial(w\tau_{zz})}{\partial z} + \rho f . V$$

Energy equation (3)

Where; $\boldsymbol{\rho}$ is density of fluid in Kg/m^3, $\boldsymbol{u}$, $\boldsymbol{v}$ and $\boldsymbol{w}$ velocities of fluid in x, y and z directions respectively in m/s, $\boldsymbol{\tau}$ is the shear stress in Pa, $\boldsymbol{t}$ is the time in s, f_x, f_y and f_z are the body forces in N, $\boldsymbol{\mu}$ is the molecular viscosity coefficient in Pa.s, $\boldsymbol{V}$ is velocity vector in m/s, $\boldsymbol{k}$ is the thermal conductivity, $\boldsymbol{e}$ is the internal energy in J , T is the temperature in C^o and $\boldsymbol{q}$ is the heat transferred *W/m²·K*.

The RNG k- ε Model

Based on a mathematical technique of renormalization group, which is proposed by [10], this model is utilized to renormalize the Navier-Stokes equations and to put the effects of smaller scales of motion into account. In contrast to the standard k-ε turbulence model, the eddy viscosity is determined from a single turbulence length scale.

a) Transport equation for kinetic energy ***k***:

$$\rho \frac{\partial k}{\partial t} = \frac{\partial}{\partial x_i}\left[\alpha_k \mu_{eff} \frac{\partial k}{\partial x_i}\right] - P_k + \rho\varepsilon \qquad (4)$$

b) Transport equation for dissipation rate **ε**:

$$\rho \frac{\partial \varepsilon}{\partial t} = \frac{\partial}{\partial x_i}\left[\alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_i}\right] - C_{1\varepsilon}\frac{\varepsilon}{k}P_k + C^*_{2\varepsilon}\frac{\varepsilon}{k}\rho\varepsilon \qquad (5)$$

Where,

$$\mu_{eff} = \mu + \mu_t \qquad (6)$$

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \qquad (7)$$

Where; $C_\mu = 0.0845, \alpha_k = \alpha_\varepsilon = 1.39, C_{1\varepsilon} = 1.42, C_{2\varepsilon} = 1.68$

The significant difference **(6)** between both Standard and RNG k-ε turbulence models is calculated from the near wall turbulence data as hereunder:

$$C^*_{2\varepsilon} = C_{2\varepsilon} - \frac{C_\mu \rho \eta^3 (1 - \frac{\eta}{\eta_o})}{1 + \beta\eta^3} \qquad (8)$$

Where;

$$\eta = \frac{k}{\varepsilon}\sqrt{2\, S_{ij}.S_{ij}} \qquad (9)$$

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \qquad (10)$$

$$\eta_o = 4.377$$

$$\beta = \; Wall\ damping, needs\ to\ be\ applied\ to\ ensure\ the\ viscosity \; = \; 0.01.$$

$\boldsymbol{\rho}$ = the density of air, $\boldsymbol{u_i}$, $\boldsymbol{u_j}$ the velocity components, ***P*** is the pressure of air in Pa, $\boldsymbol{\mu}$ is the dynamic viscosity, ***t*** is the time, ***i***, ***j*** are 1,2,3.

Experimental Part

Description of Baghdad Tower as Real Case Study

A building, which is known “Baghadad tower" is a large building tower located in Baghadad city beside of Al-nusur Square. The Baghdad tower is a 205-meter high TV tower as shown in the (**Figure 1**). The Baghadad tower was opened in 1994 and replaced the television tower that was destroyed during the Gulf War. On the top floor, there is a revolving restaurant. The height of the Baghdad tower used in this study is 150 meters. Part of the column above the tower was neglected. The wind tunnel scale was chosen to be 1:250 as that is typical in industry practice to use this factor in studies. In the case of the chosen building, this reduced it is to 600 mm high as seen in (**Figure 2**). The experimental model is manufactured from wood. The wood plate connection seams were filled with wood putty to ensure smooth wind flow over the seams. The model is open from the base for entering the tubes of measuring air pressure measuring tubes. Seven pressure ports are made along the height of building tower model for measurement of wind pressures and are shown in (**Figure 2**). Wind direction on model as shown in the (**Figure 3**). The pressure ports were placed on equal distance on the tower building, and the distance between the pressures ports were 60 mm. The laboratory-building tower model that has been used in experimental is shown in (**Figure 4**).

Figure (1) Real Baghdad Tower.

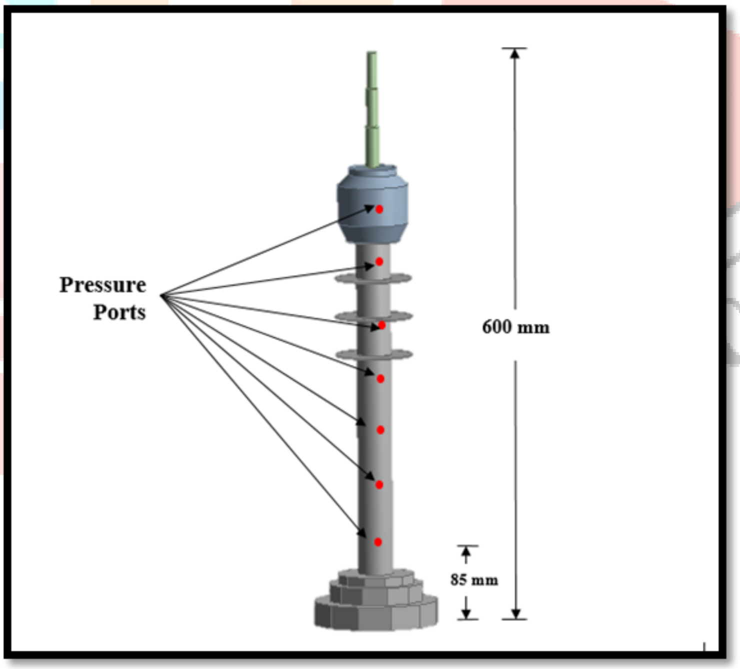


Figure (2) The Height and Pressure Ports of Baghdad Tower Model.

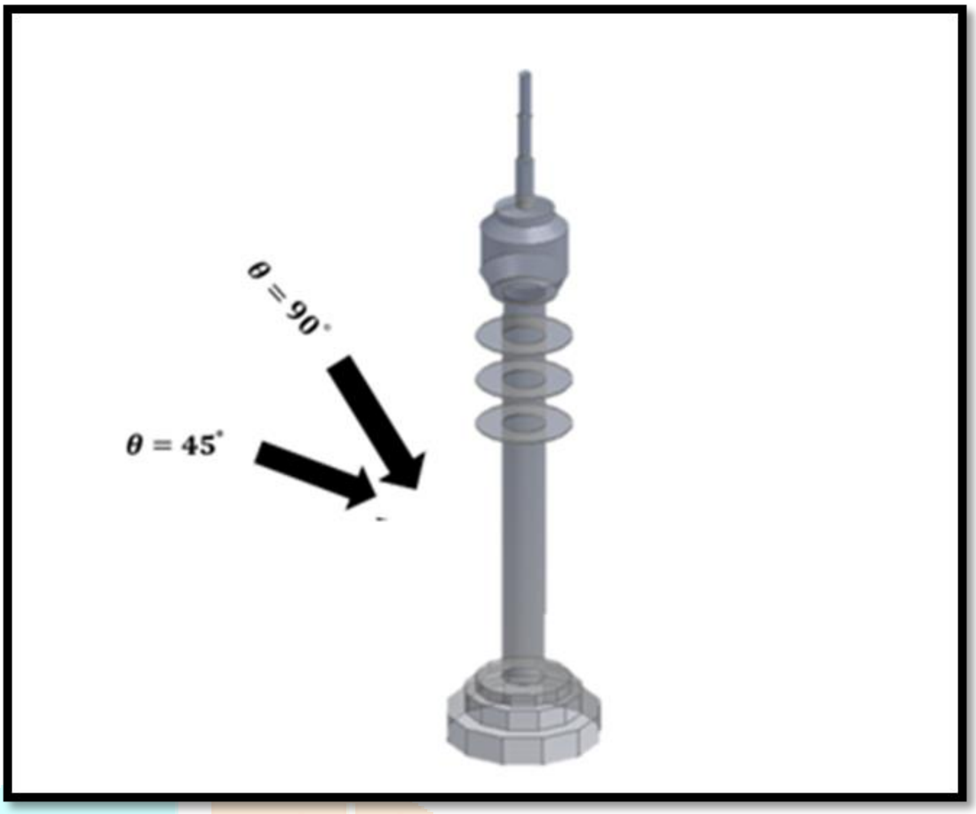


Figure (3) Wind Directions on Model.

Figure (4) Photographs Showing the Laboratory Horizontal Wind Tunnel Baghdad Tower Model.

Description of Baghdad University Tower as Real Case Study

The building, known as the Baghdad University tower, is one of the tallest buildings in the capital, Baghdad University, near the Jadriyah Bridge. Designed by the German (Walter Kropis, 1957) as shown in the (**Figure 5**). Baghdad University Tower consists of 20 floors. The dimensions of the tower are (27.45 x 24.69 x 82.74) meters. The wind tunnel scale was chosen to be 1:140 as that is typical in industry practice to use this factor in studies. In the case of the chosen scale for building, the dimensions became (196 x 176 x 591) mm as seen in (**Figure 6**). The experimental model is manufactured from wood. The wood plate connection seams were filled with wood putty to ensure smooth wind flow over the seams. The model is open from the base for in order to allow the introduction of tubes to measure air pressure measuring tubes. Eighteen pressure ports are made on the each faces of tower model building for measurement of wind pressures as are shown in (**Figure 6**) and the locations of pressure ports on faces of tower model are shown in the **Table (1)** and **(2)**. The number and location of pressure ports for opposite faces for tower model are same. Wind direction on model as shown in the (**Figure 7**). The laboratory-building tower model that has been used in experimental is shown in (**Figure 8**).

Figure (5) Real Baghdad University Tower.

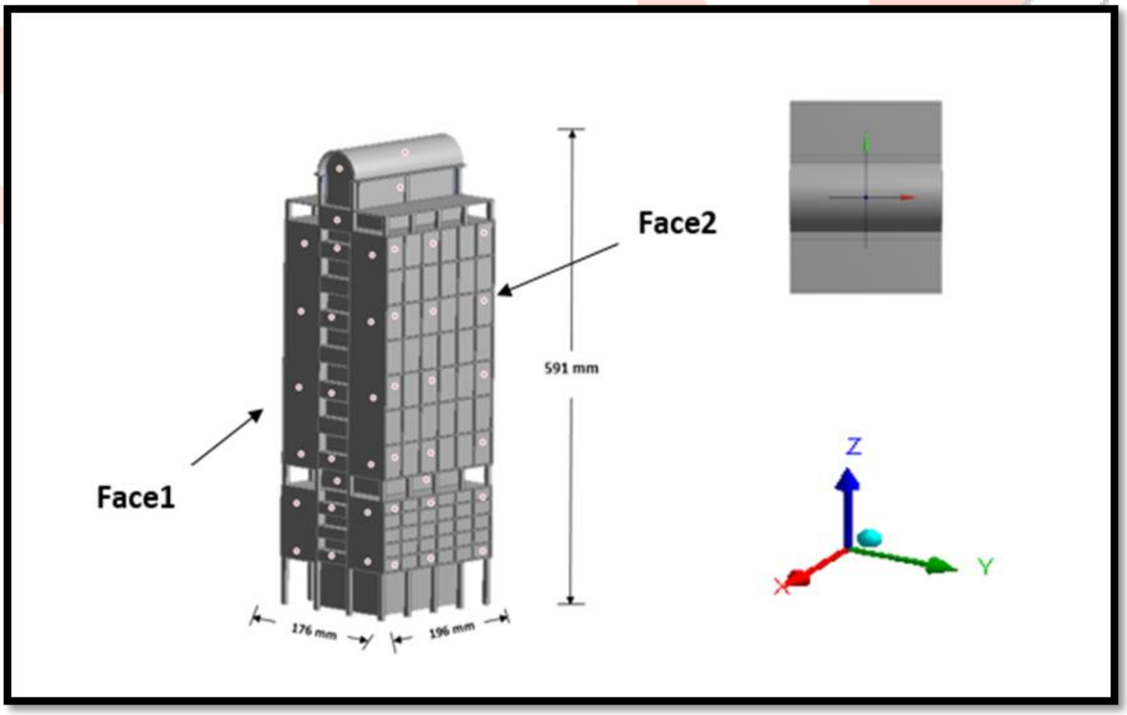


Figure (6) Dimensions and Pressure Ports of Baghdad University Tower Model.

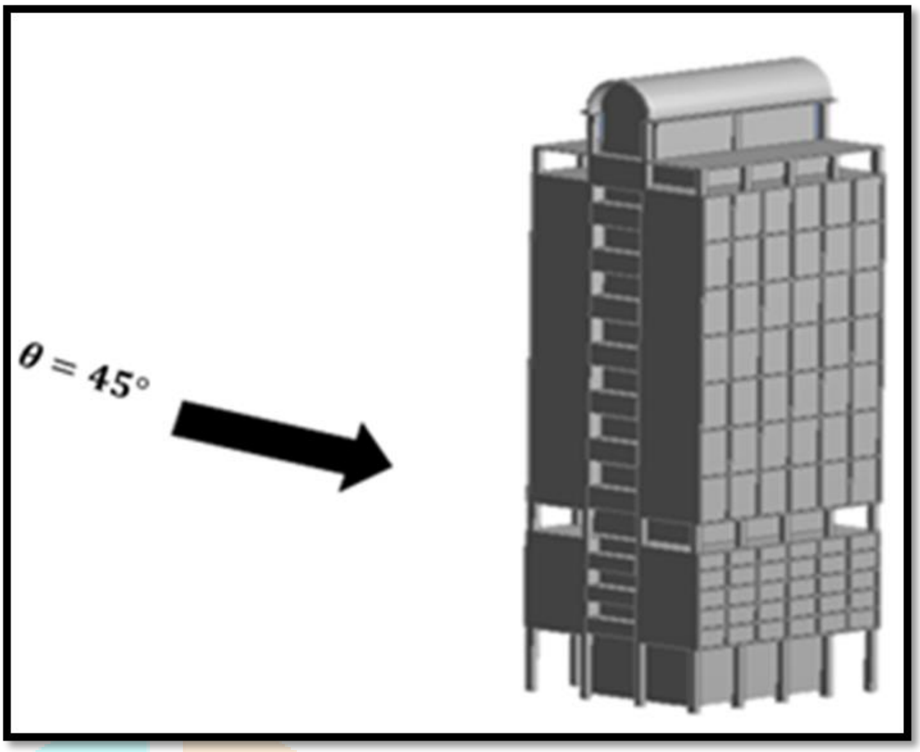


Figure (7) Wind Directions on Model.

Table (1) Pressure Port Numbers and Locations for Face1 of Baghdad University Tower Model.

Point Number	X [mm]	Y [mm]	Z [mm]
1	98	-70	70
2	98	0	70
3	98	70	70
4	98	-70	135
5	98	0	135
6	98	70	135
7	75	0	170
8	98	-70	240
9	98	0	240
10	98	70	240
11	98	-70	375
12	98	0	375
13	98	70	375
14	98	-70	470
15	98	0	470
16	98	70	470
17	98	0	505
18	90	0	570

Table (2) Pressure Port Numbers and Locations for Face2 Baghdad University Tower Model.

Point Number	X [mm]	Y [mm]	Z [mm]
1	-85	88	70
2	-10	88	70
3	85	88	70
4	-85	88	145
5	-10	88	145
6	85	88	145
7	-10	60	170
8	-85	88	250
9	-10	88	250
10	85	88	250
11	-85	88	385
12	-10	88	385
13	85	88	385
14	-85	88	475
15	-10	88	475
16	85	88	475
17	-10	60	505
18	0	35	580

Figure (8) Photographs Showing the Laboratory Horizontal Wind Tunnel Baghdad University Tower Model.

This low speed open circuit wind tunnel has been designed, manufactured and constructed at the Mechanical Engineering Department at Baghdad University - College of Engineering. The work is one of the pioneer projects adapted by the R & D Office at the Iraqi MOHESR. The flow cross-section WxH: 700x700 mm and with length 1500 mm as shown in (**Figure 9**). The wind velocity was varying from 1 to 70 m/s.

The bed and sidewalls of wind tunnel are made from wood. Every model is placed in the wind tunnel at the center of the measurement section. The wind velocity used in the test is 30, 40 and 50 m/s.

Figure (9) Photographs Showing the Laboratory Wind Tunnel.

CFD Simulation Part

The optimum mesh sizes that selected to be adequately modeled the wind flow over two building models were having 1 mm for edges and 5 mm for face. The boundary conditions of the two building models are shown (**Figure 10**). The unstructured mesh scheme is applied in this paper as shown (**Figure 11**). The number of cells for Baghdad tower model is about $300x10^3$ and for Baghdad University tower is about $11x10^5$. The number of iteration is 5000.

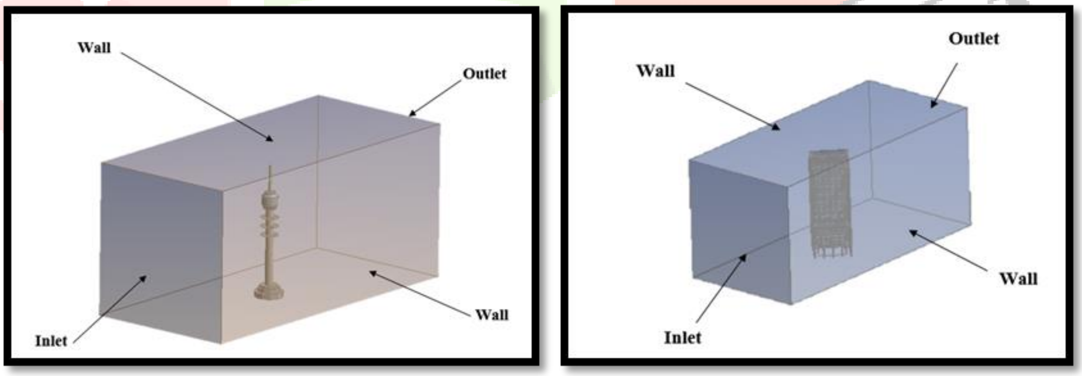
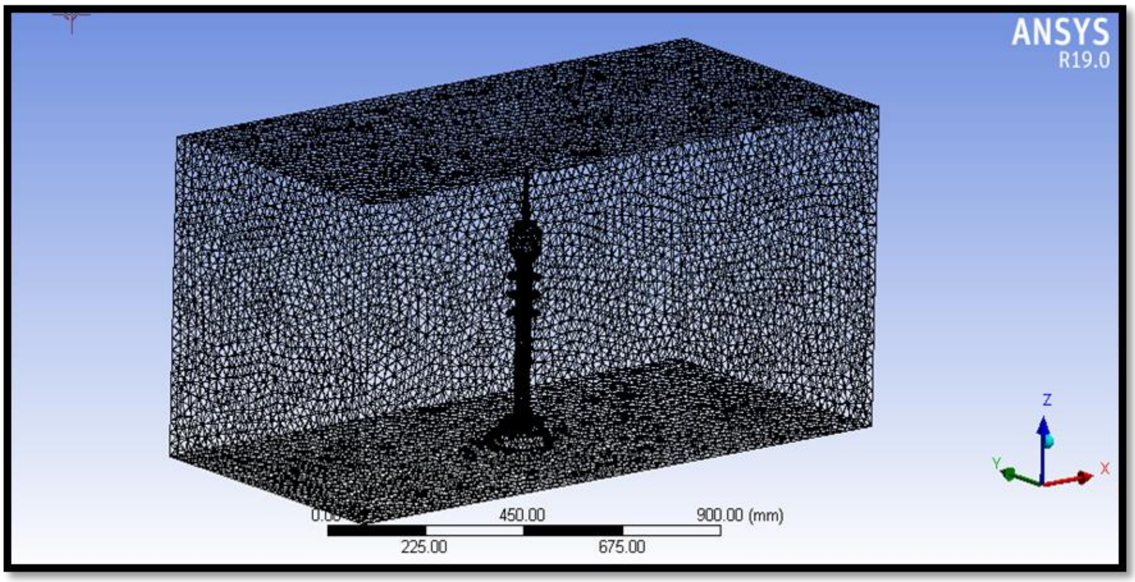


Figure (10) the Boundary Conditions of two Models Study.



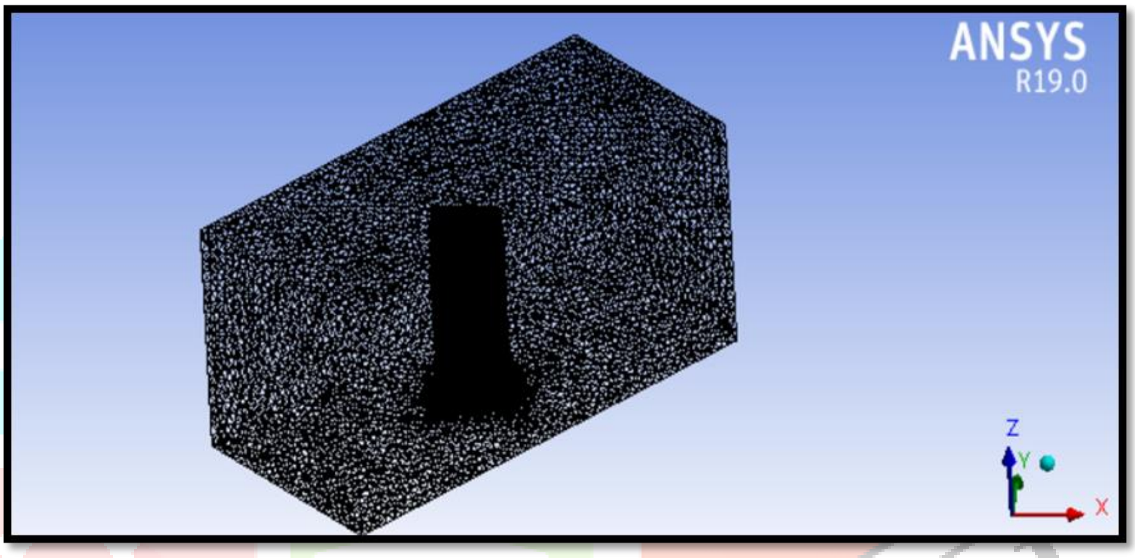


Figure (11) Mesh Generation.

Pressure Results and Discussion for Baghdad Tower

Figure (12), **(14), (16), (18), (20) and (22)** indicated the results of pressure for sideward1, while **Figure (13)**, **(15), (17)**, **(19)**, **(21)** and **(23)** shown the results of pressure for sideward2 for different value of wind velocity (30, 40 and 50 m/s). **Figure (24)** shown the pressure contour and velocity streamline results for model. Relative errors for the pressure between numerical and experimental results are generally For Baghdad tower model is 0.11% to 5.78%. It can be seen from **Figure (24)** of velocity distribution for model that the speed on the practical side is the same on the theoretical side and starts to change when exposed to friction. It can be indicated that the wind distribution on the model shows a small change in speed as the shape of the cylinder decrease friction and makes the change in speed very small. This is an advantage that the Baghdad Tower has over the rest of the high buildings in Iraq.

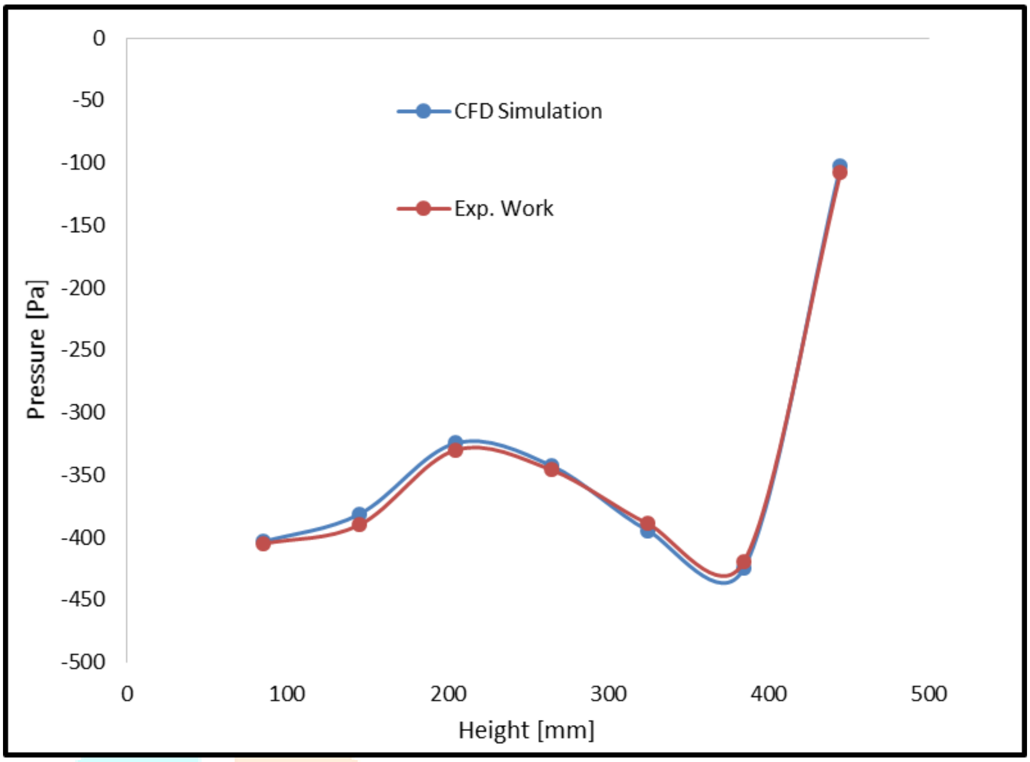


Figure (12) Comparison between Pressures Obtained by Experimental and Numerical Models for Sideward1 Model for Wind Velocity 30 m/s and 45º Angle of Wind Attack.

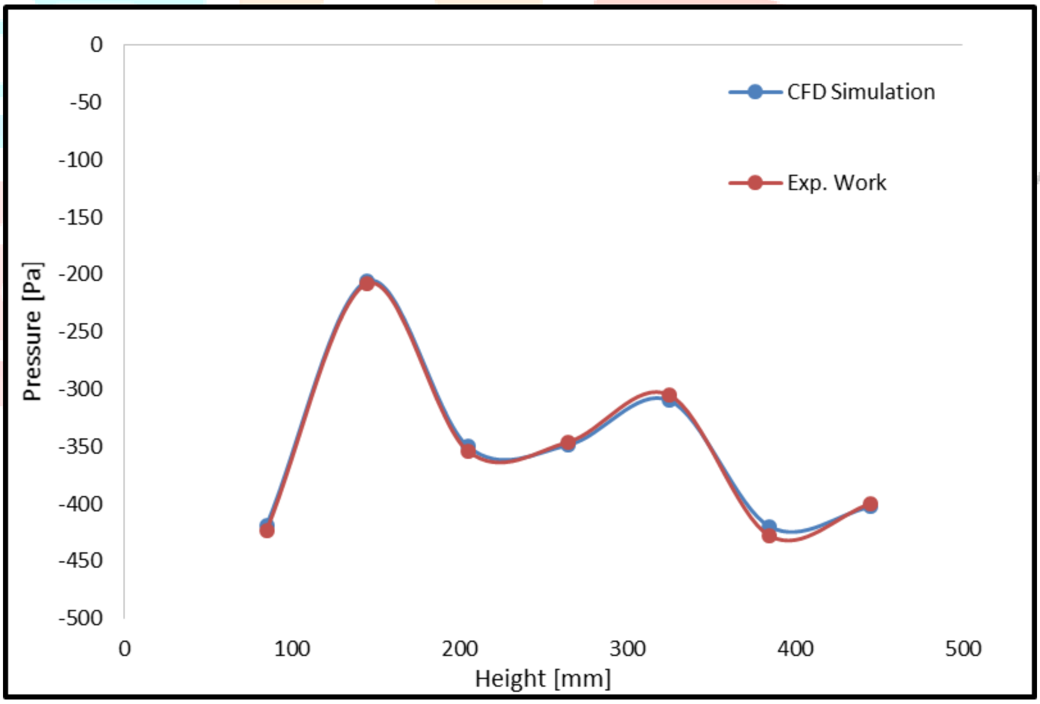


Figure (13) Comparison between Pressures Obtained by Experimental and Numerical Models for Sideward2 Model for Wind Velocity 30 m/s and 45º Angle of Wind Attack.

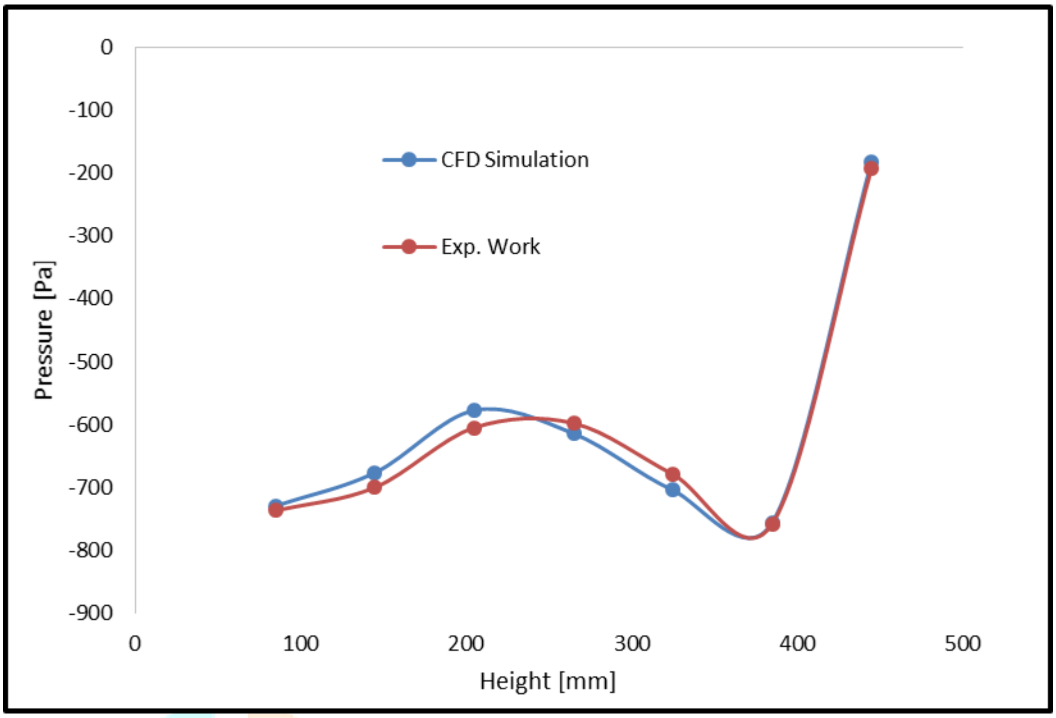


Figure (14) Comparison between Pressures Obtained by Experimental and Numerical Models for Sideward1 Model for Wind Velocity 40 m/s and 45º Angle of Wind Attack.

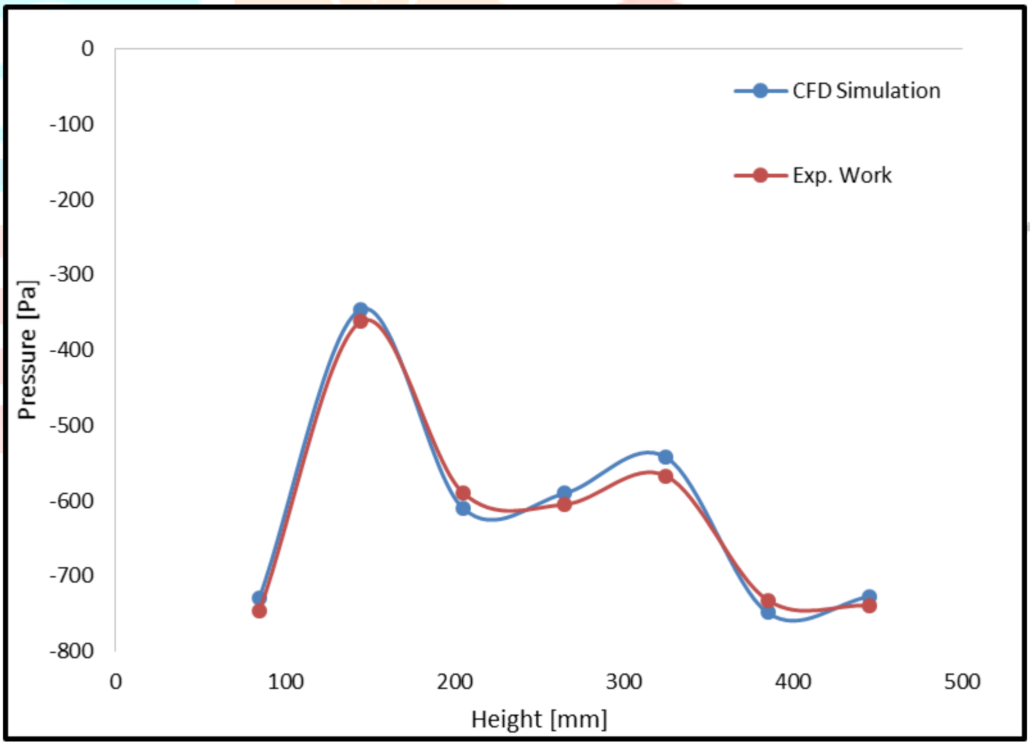


Figure (15) Comparison between Pressures Obtained by Experimental and Numerical Models for Sideward2 Model for Wind Velocity 40 m/s and 45º Angle of Wind Attack.

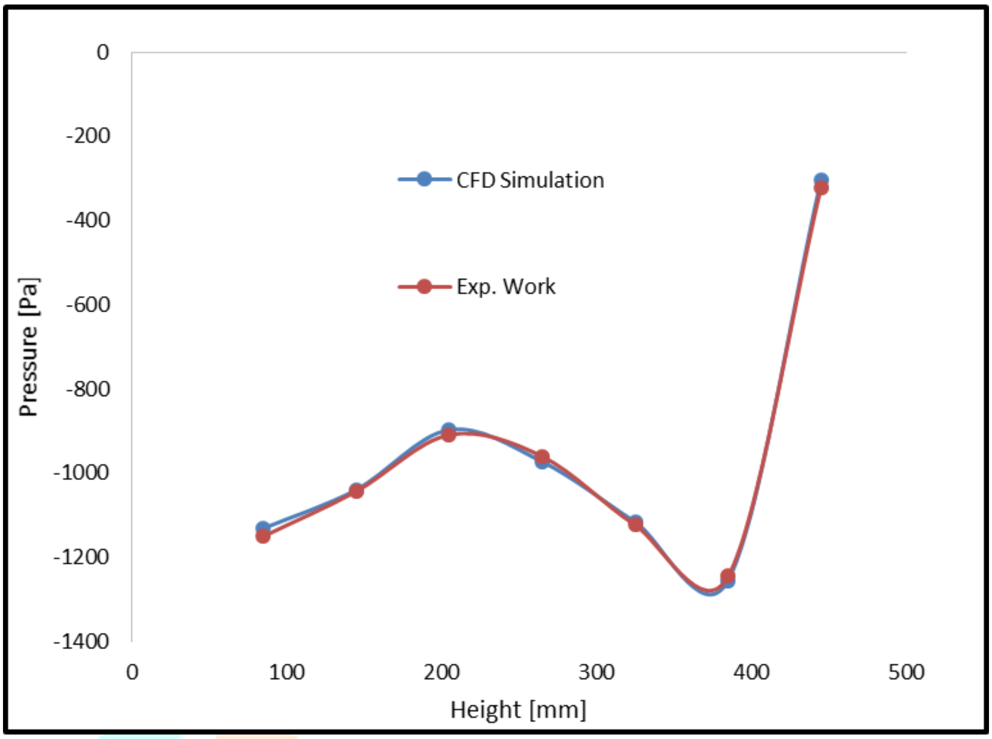


Figure (16) Comparison between Pressures Obtained by Experimental and Numerical Models for Sideward1 Model for Wind Velocity 50 m/s and 45º Angle of Wind Attack.

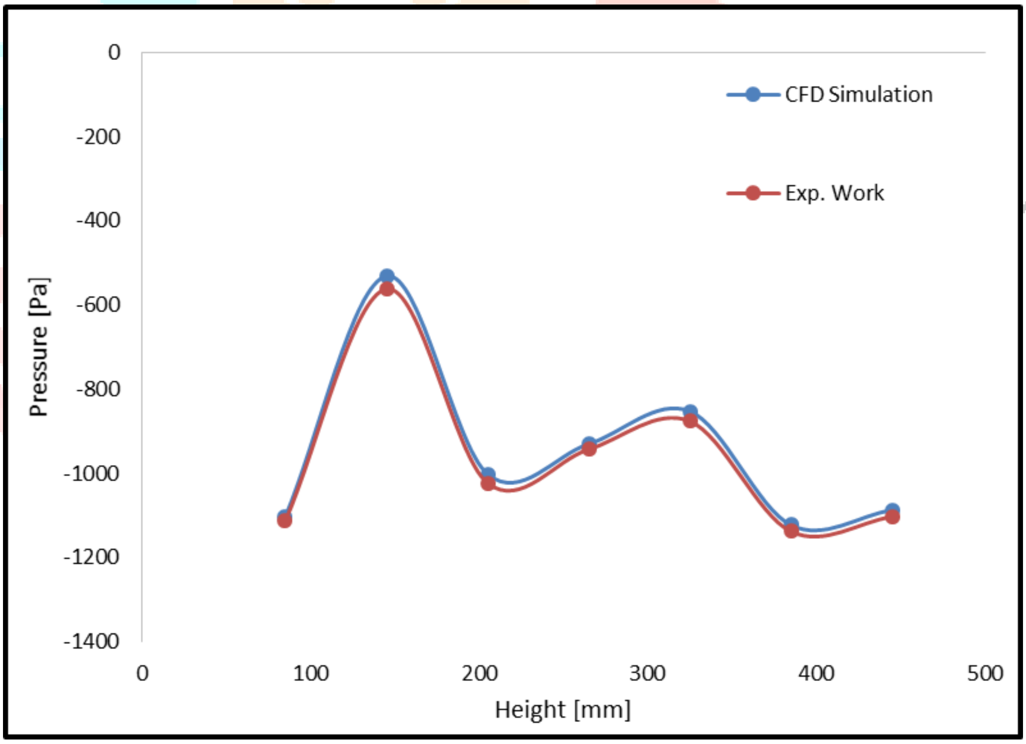


Figure (17) Comparison between Pressures Obtained by Experimental and Numerical Models for Sideward2 Model for Wind Velocity 50 m/s and 45º Angle of Wind Attack.

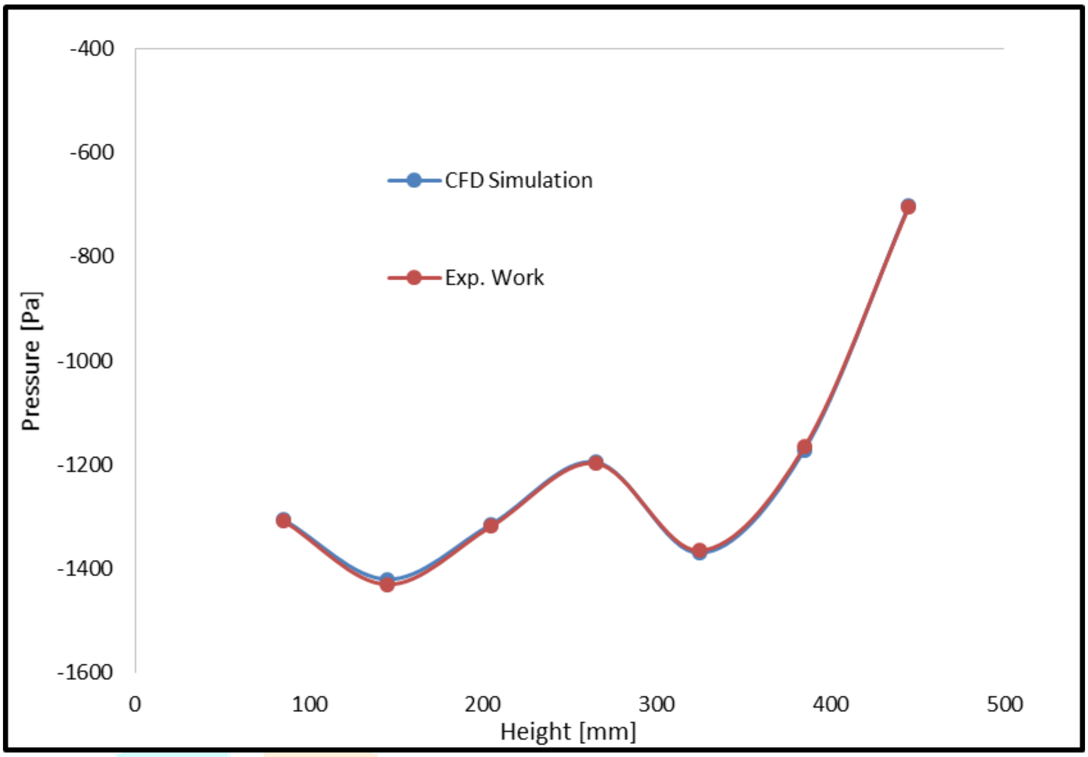


Figure (18) Comparison between Pressures Obtained by Experimental and Numerical Models for Sideward1 Model for Wind Velocity 30 m/s and 90º Angle of Wind Attack.

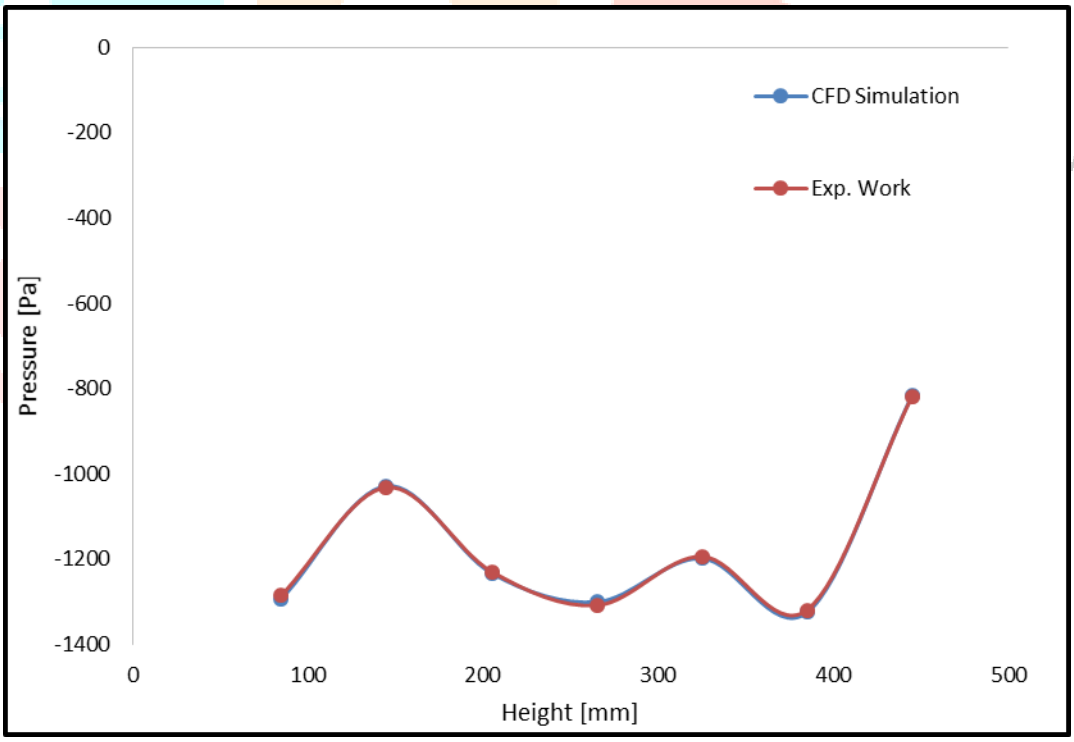


Figure (19) Comparison between Pressures Obtained by Experimental and Numerical Models for Sideward2 Model for Wind Velocity 30 m/s and 90º Angle of Wind Attack.

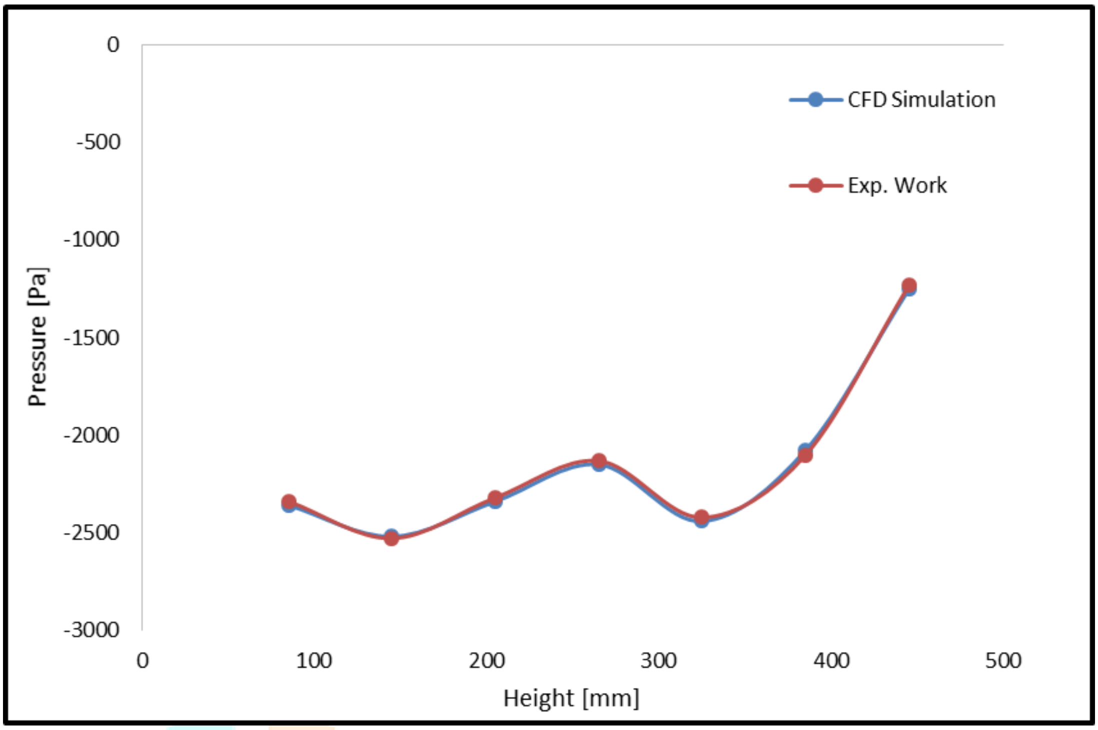


Figure (20) Comparison between Pressures Obtained by Experimental and Numerical Models for Sideward1 Model for Wind Velocity 40 m/s and 90º Angle of Wind Attack.

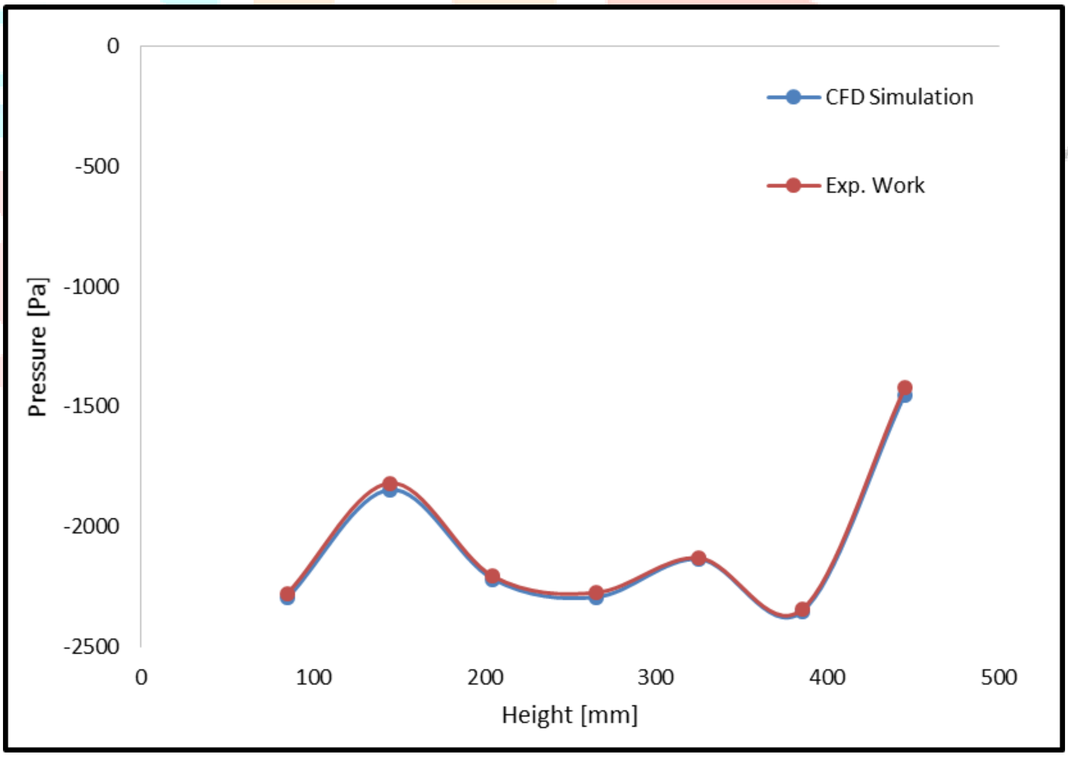


Figure (21) Comparison between Pressures Obtained by Experimental and Numerical Models for Sideward2 Model for Wind Velocity 40 m/s and 90º Angle of Wind Attack.

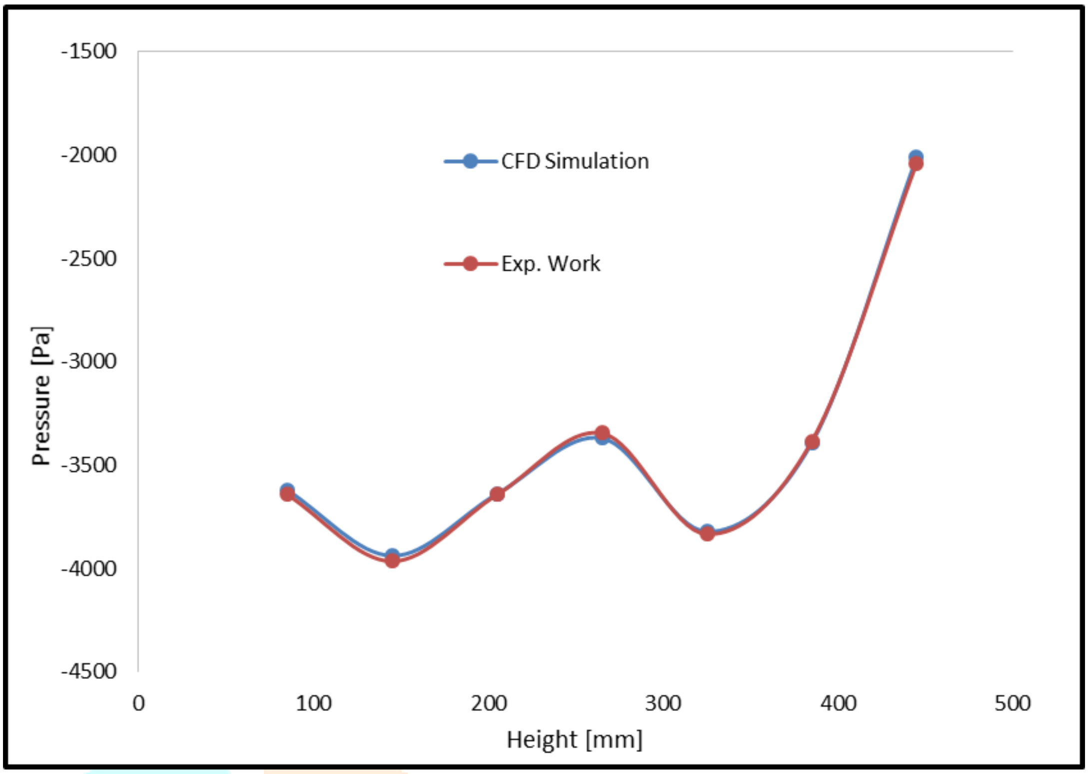


Figure (22) Comparison between Pressures Obtained by Experimental and Numerical Models for Sideward1 Model for Wind Velocity 50 m/s and 90º Angle of Wind Attack.

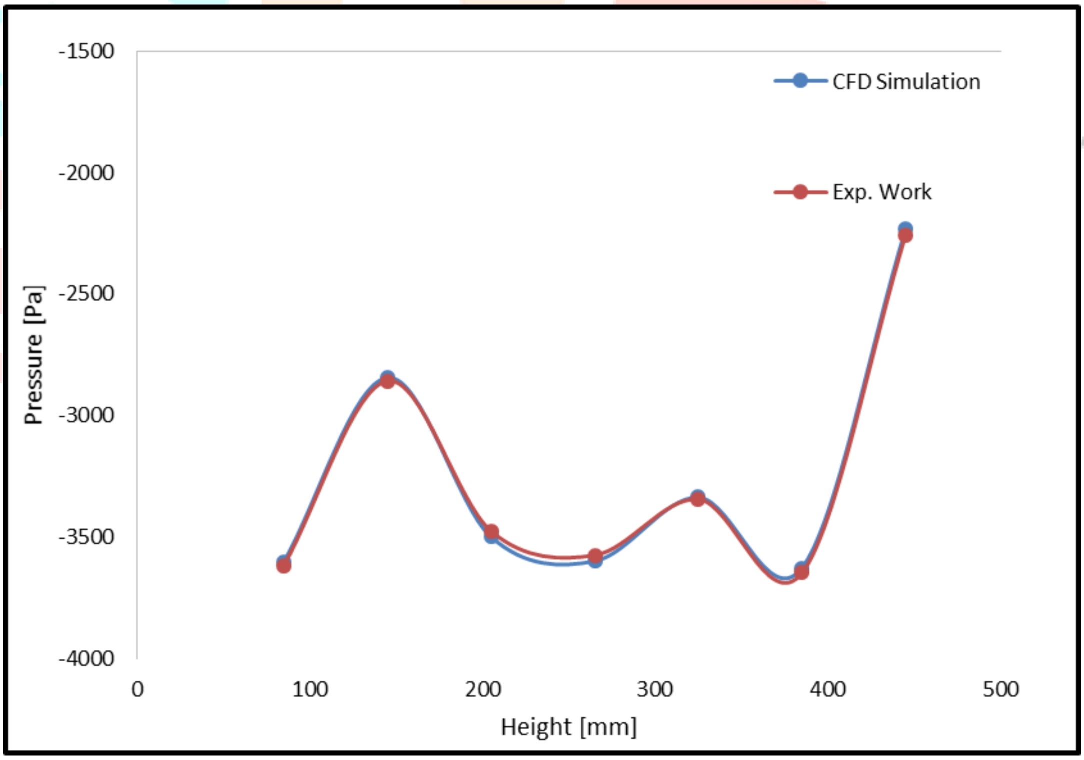


Figure (23) Comparison between Pressures Obtained by Experimental and Numerical Models for Sideward2 Model for Wind Velocity 50 m/s and 90º Angle of Wind Attack.

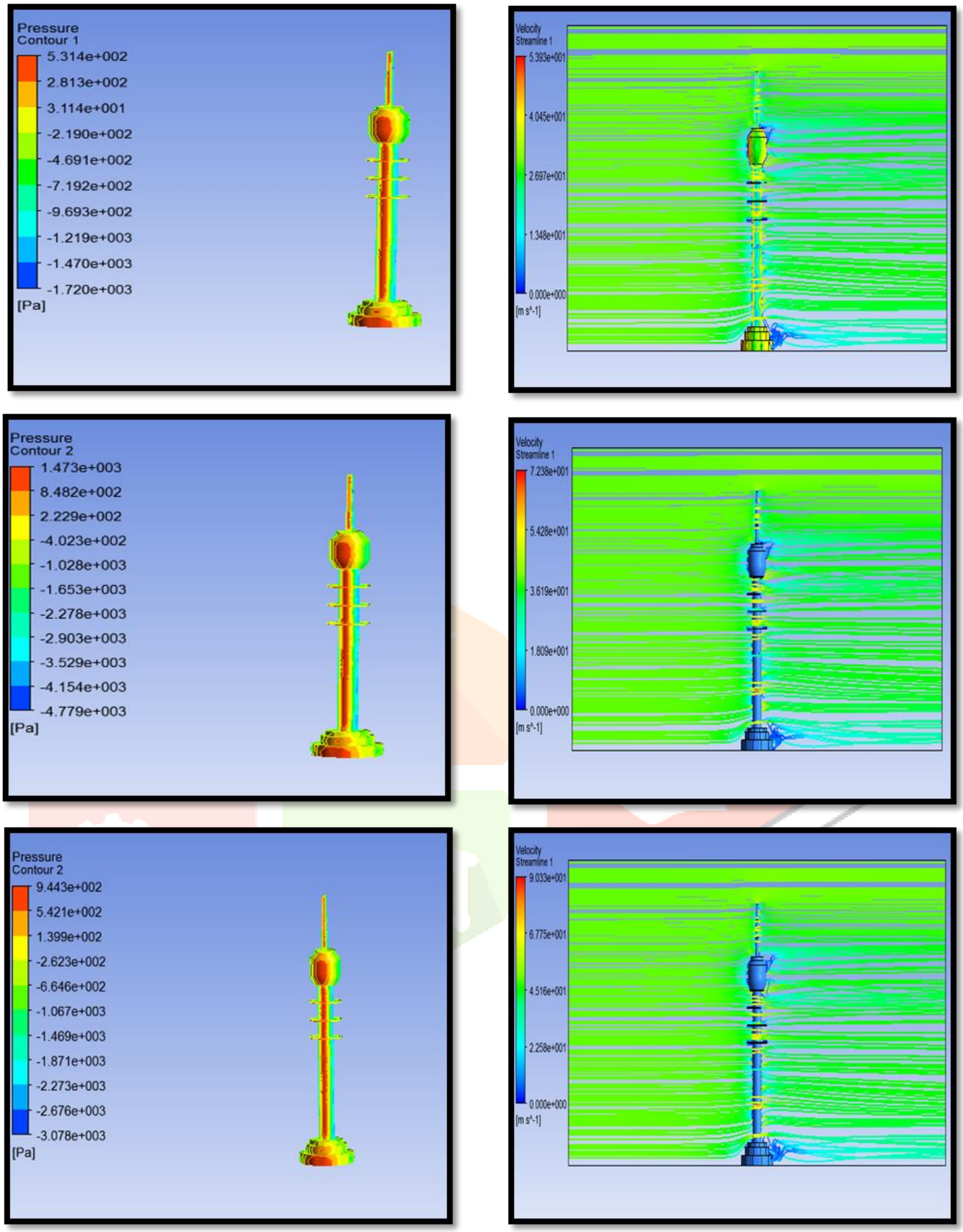


Figure (24) Contour and Streamline Results for Baghdad Tower Model for Deferent Velocity.

Pressure Results and Discussion for University Baghdad Tower

Figure (25), (29) and **(33)** indicated the results of pressure for windward1, while **Figure (26), (30)** and **(34)** shown the results of pressure for leeward1, while **Figure (27), (31)** and **(35)** indicated the results of pressure for windward2, while **Figure (28), (32)** and **(36)** shown the results of pressure for leeward2 with angle of wind 45 degree. All these for different value of wind velocity (30, 40 and 50 m/s). **Figure (37)** shown the pressure contour and velocity streamline results for model. Relative errors for the pressure between numerical and experimental results are generally for University Baghdad tower model is 0.18% to 6.17%. It can be seen from **Figure (37)** of velocity streamline distribution for model that the wind speed on the real side is the same on the hypothetical side and starts to change when exposed to resistance. It can be showed that the wind distribution on the University Baghdad tower model shows a strong change in speed as the shape of the tower increases friction with wind and makes the change in speed so large.

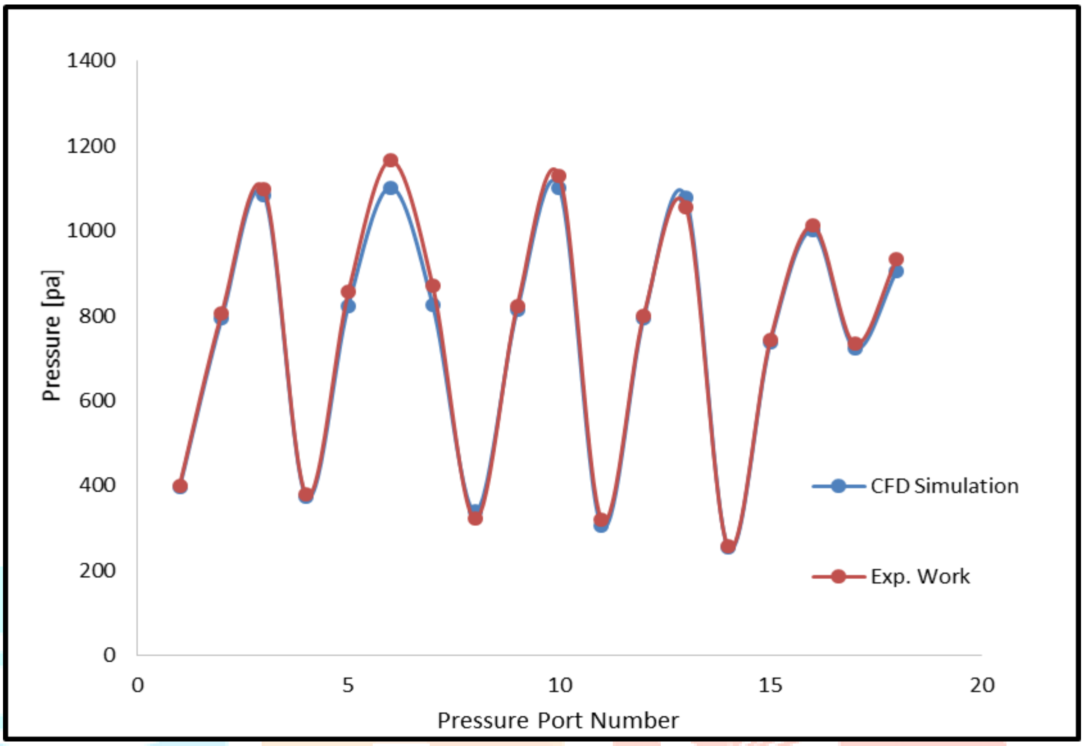


Figure (25) Comparison between Pressures Obtained by Experimental and Numerical Models for Windward1 Model for Wind Velocity 30 m/s and 45° Angle of Wind Attack.

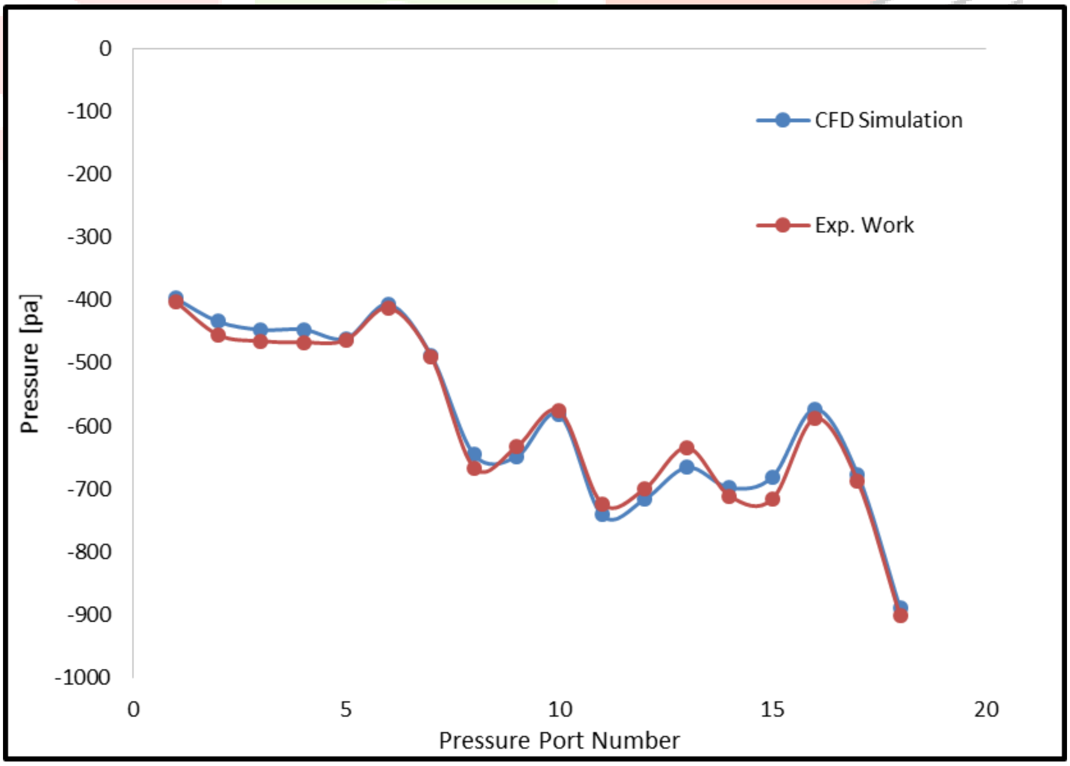


Figure (26) Comparison between Pressures Obtained by Experimental and Numerical Models for Leeward1 Model for Wind Velocity 30 m/s and 45° Angle of Wind Attack.

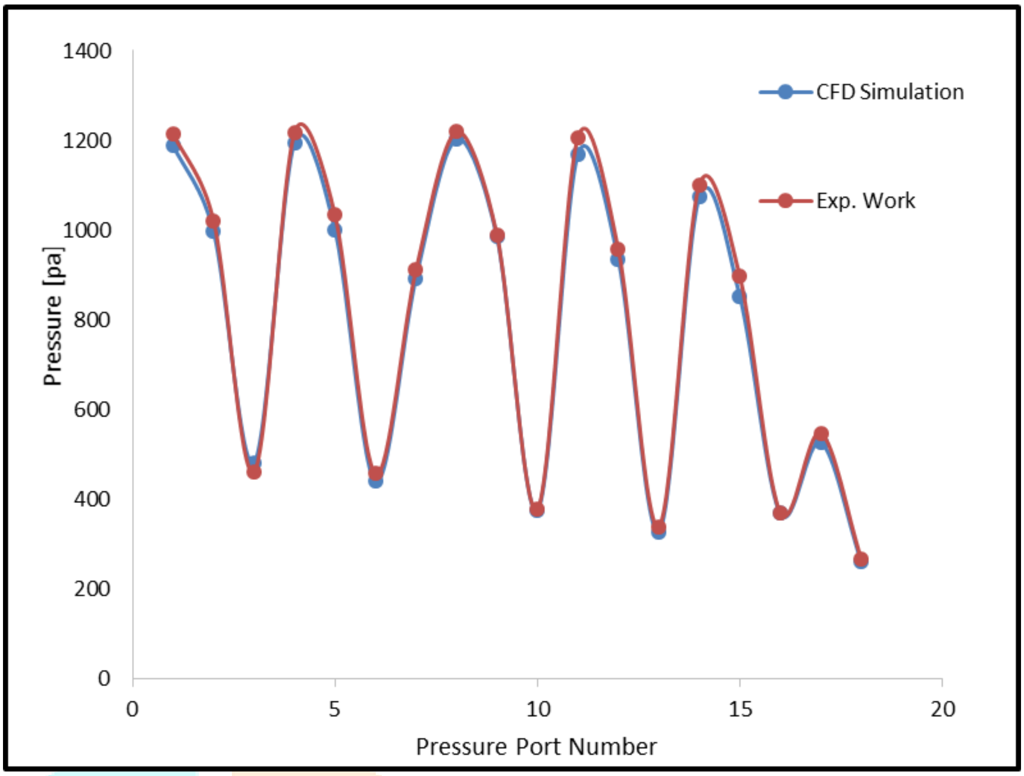


Figure (27) Comparison between Pressures Obtained by Experimental and Numerical Models for Windward2 Model for Wind Velocity 30 m/s and 45° Angle of Wind Attack.

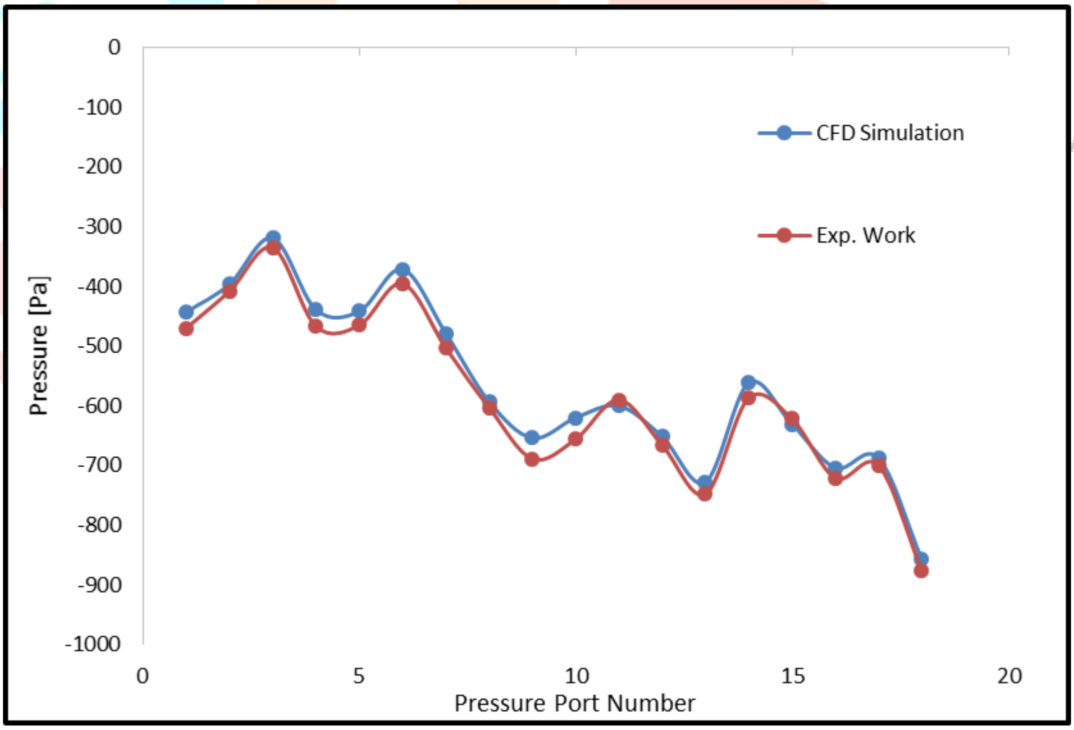


Figure (28) Comparison between Pressures Obtained by Experimental and Numerical Models for Leeward2 Model for Wind Velocity 30 m/s and 45° Angle of Wind Attack.

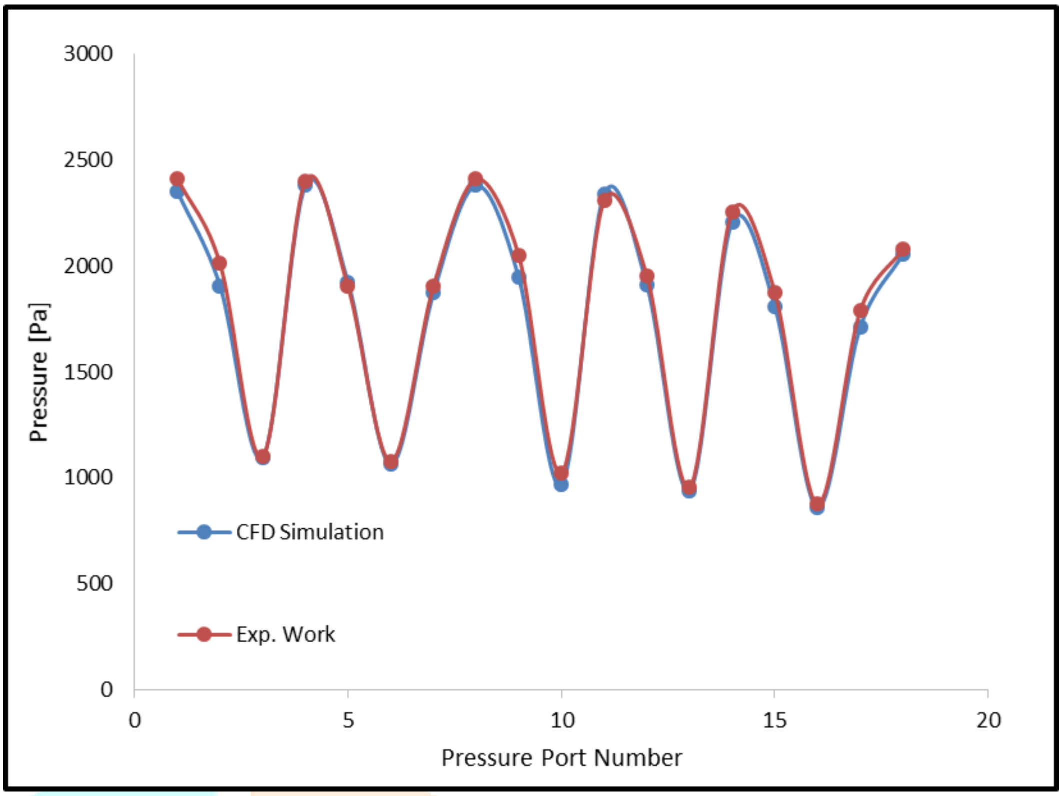


Figure (29) Comparison between Pressures Obtained by Experimental and Numerical Models for Windward1 Model for Wind Velocity 40 m/s and 45° Angle of Wind Attack.

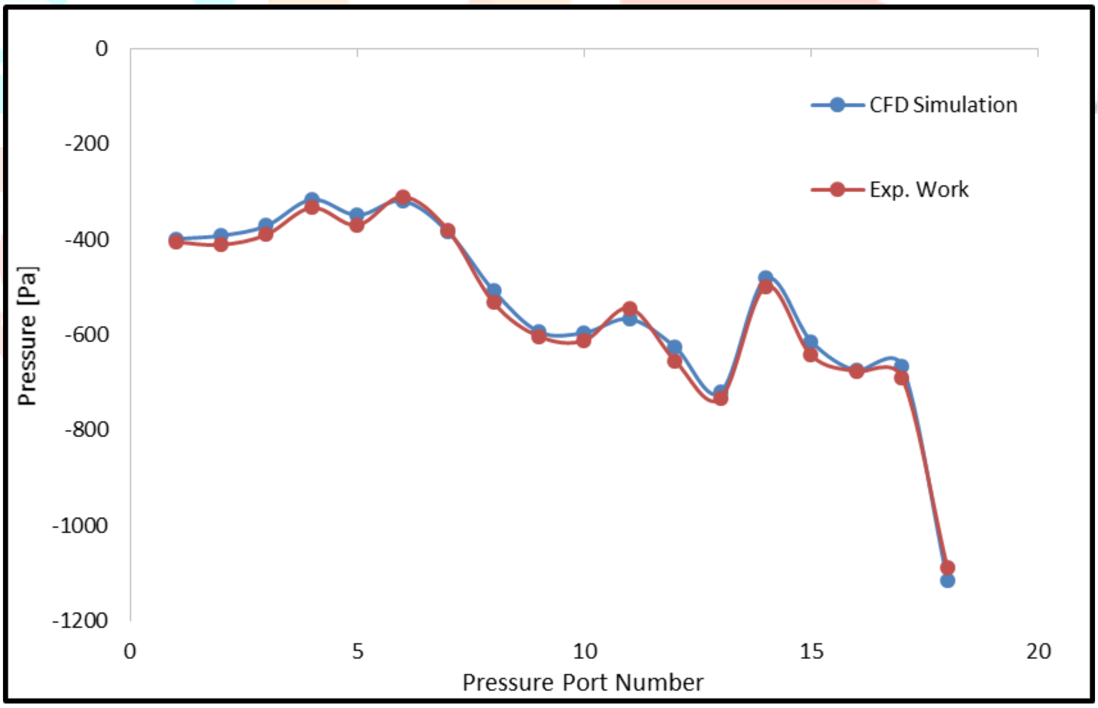


Figure (30) Comparison between Pressures Obtained by Experimental and Numerical Models for Leeward1 Model for Wind Velocity 40 m/s and 45° Angle of Wind Attack.

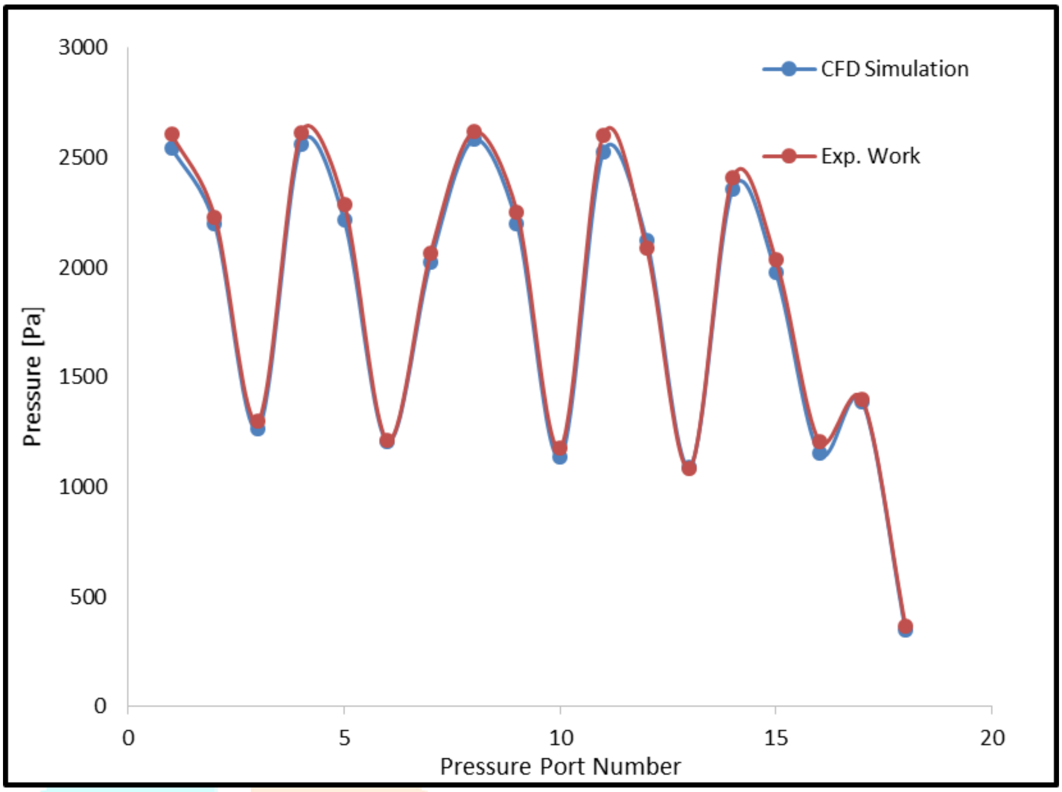


Figure (31) Comparison between Pressures Obtained by Experimental and Numerical Models for Windward2 Model for Wind Velocity 40 m/s and 45° Angle of Wind Attack.

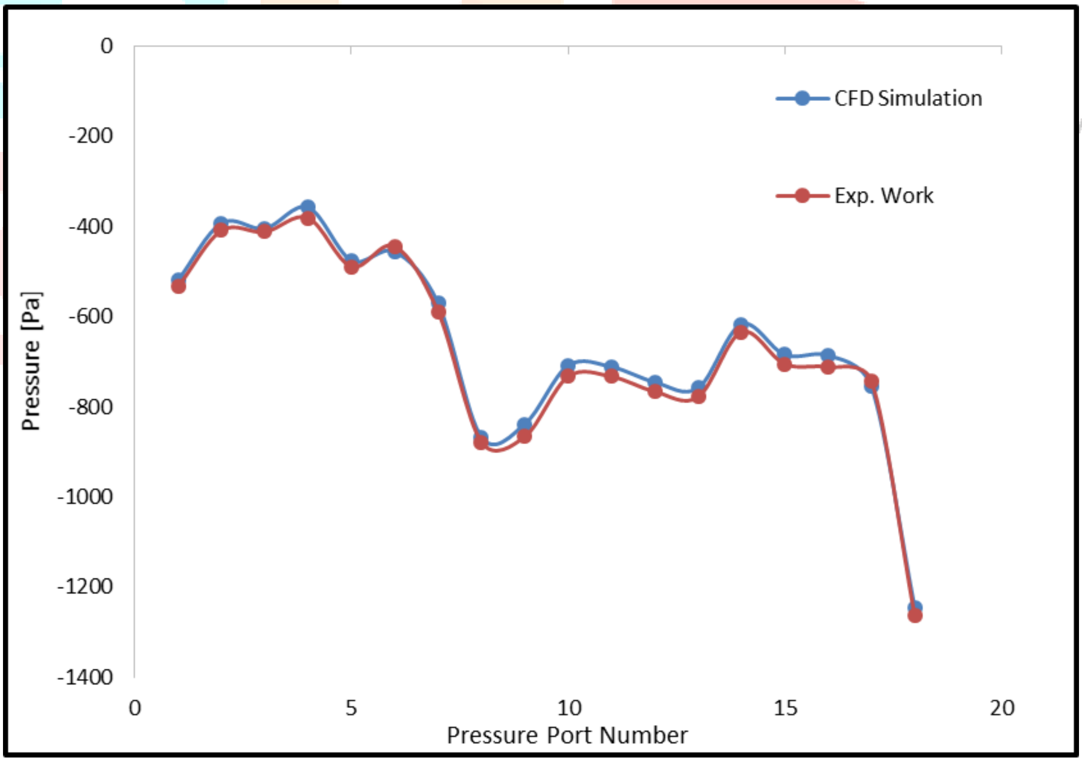


Figure (32) Comparison between Pressures Obtained by Experimental and Numerical Models for Leeward2 Model for Wind Velocity 40 m/s and 45° Angle of Wind Attack.

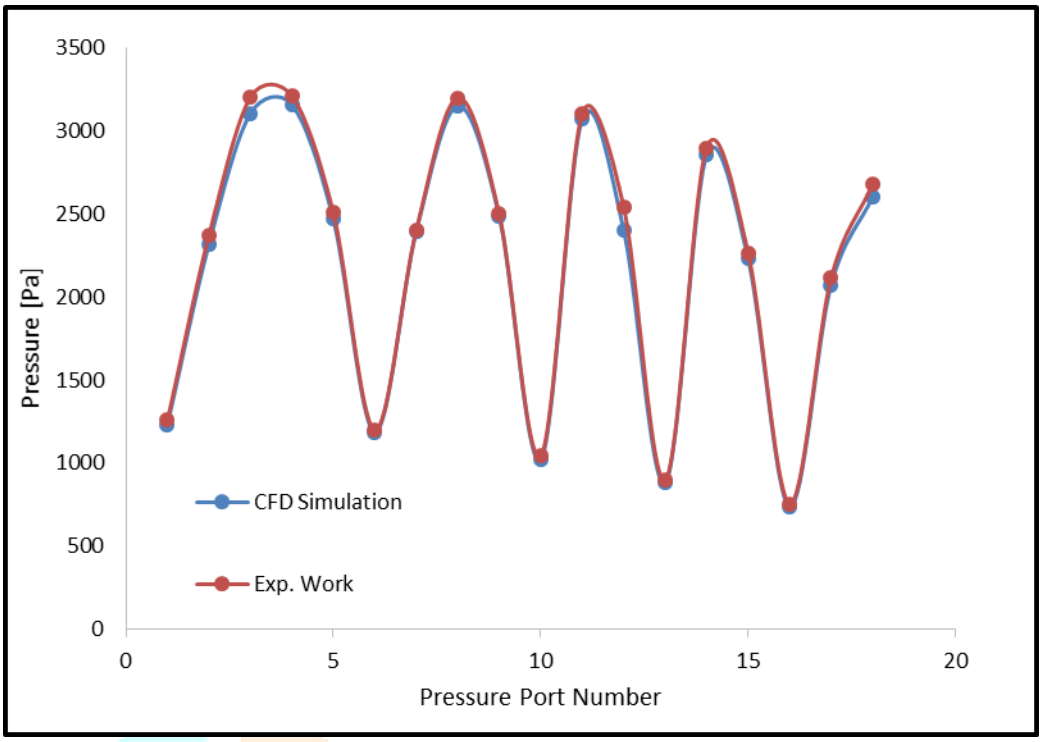


Figure (33) Comparison between Pressures Obtained by Experimental and Numerical Models for Windward1 Model for Wind Velocity 50 m/s and 45° Angle of Wind Attack.

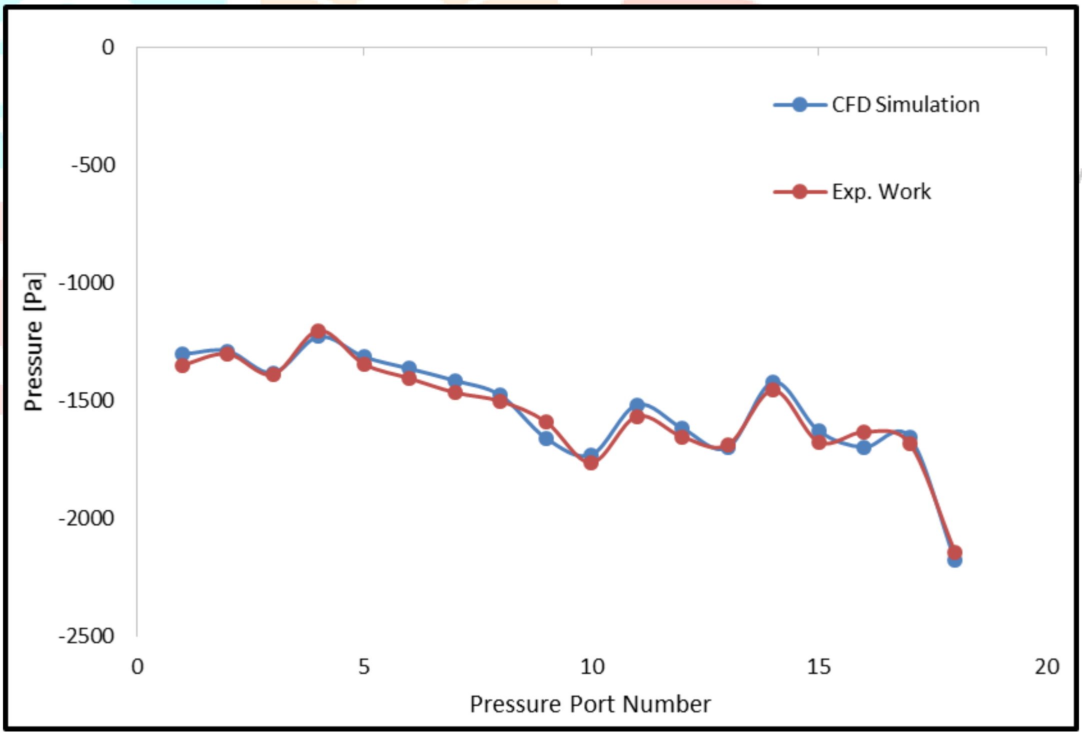


Figure (34) Comparison between Pressures Obtained by Experimental and Numerical Models for Leeward1 Model for Wind Velocity 50 m/s and 45° Angle of Wind Attack.

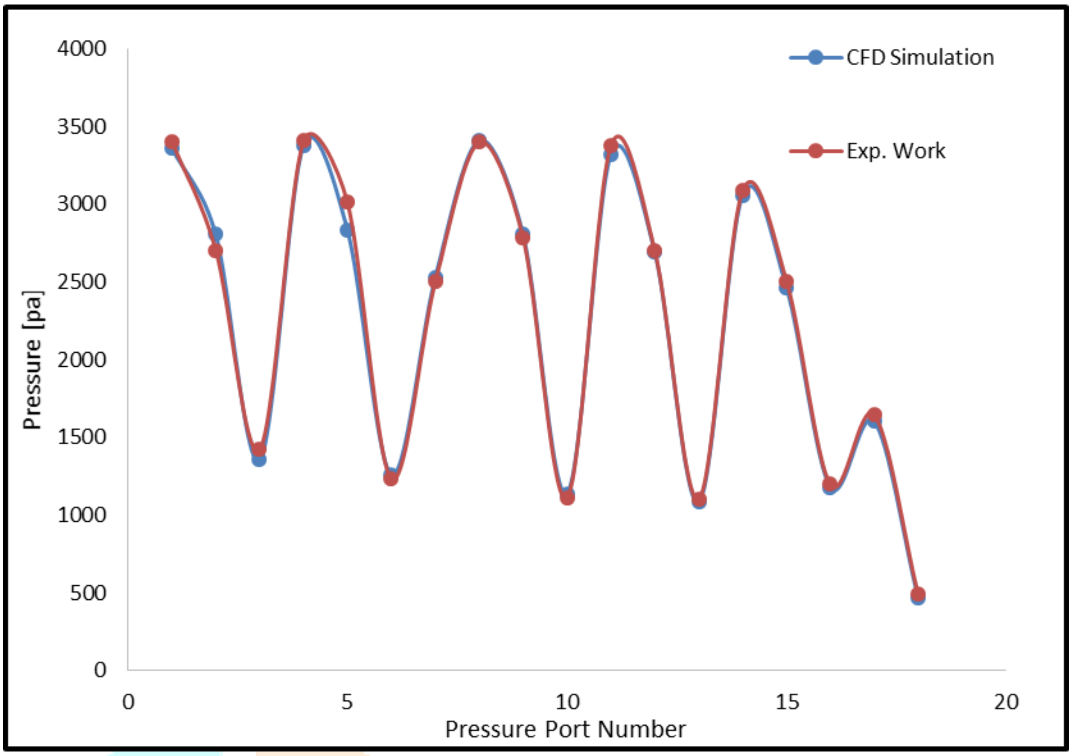


Figure (35) Comparison between Pressures Obtained by Experimental and Numerical Models for Windward2 Model for Wind Velocity 50 m/s and 45° Angle of Wind Attack.

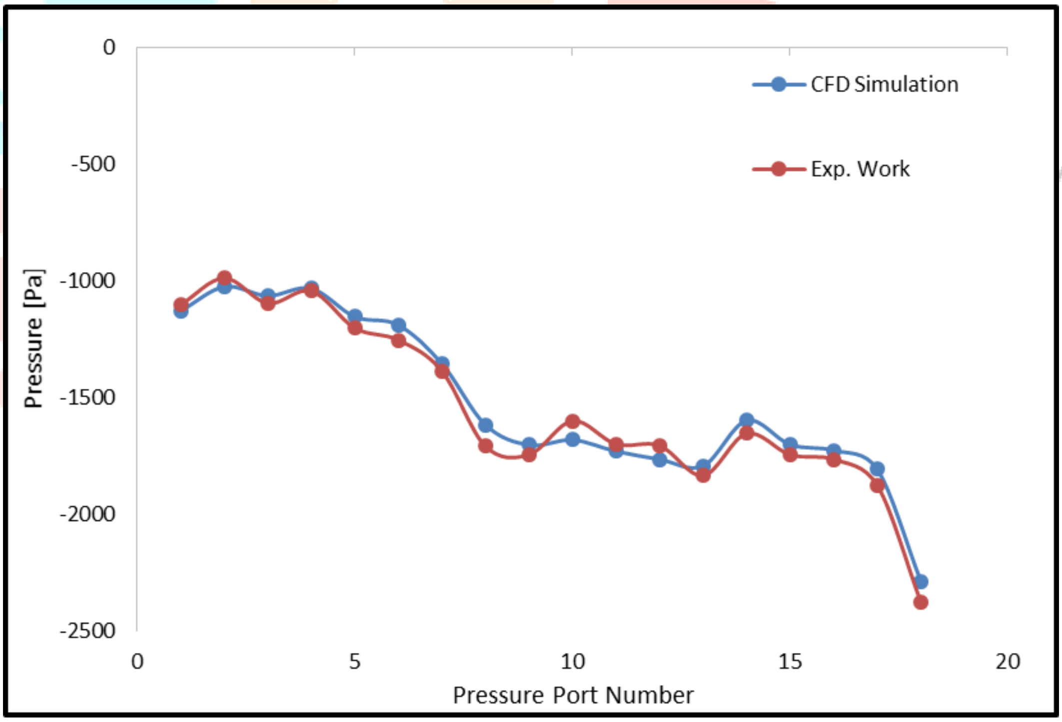


Figure (36) Comparison between Pressures Obtained by Experimental and Numerical Models for Leeward2 Model for Wind Velocity 50 m/s and 45° Angle of Wind Attack.

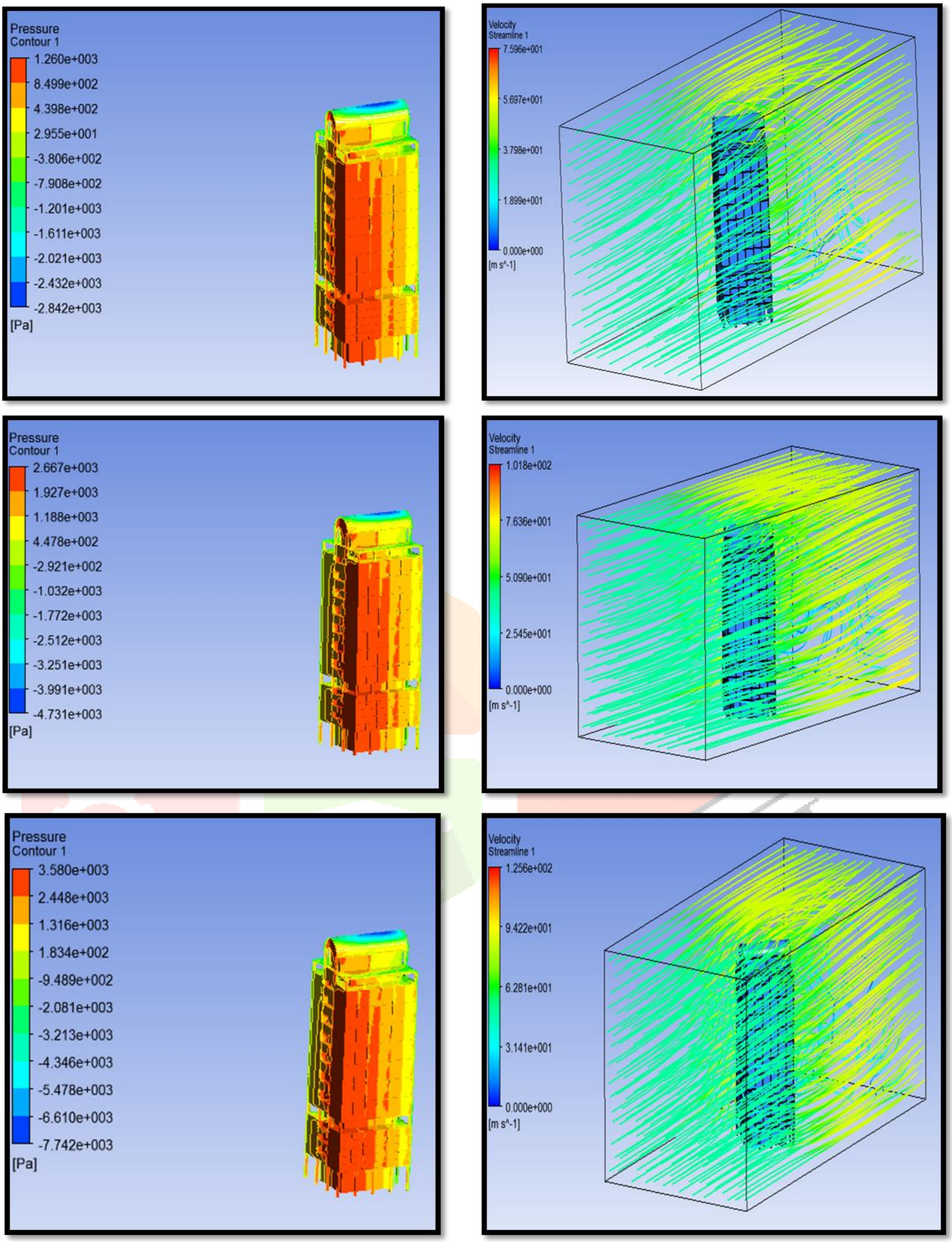


Figure (37) Contour and Streamline Results for Baghdad University Tower Model for Deferent Velocity.

Design Pressure Comparison between Two Models and Effect of Models Shape and Wind angles on Result

Based on numerical results, the design pressure is calculated for the second two models. (**Figure 38)** shows a comparison of the design pressure results of the second two models with three different velocities. It can be concluded from this comparison that the design pressure of the Baghdad tower model is less than the design pressure of the Baghdad University tower model with different speed with the percentage difference is about (50-60%).

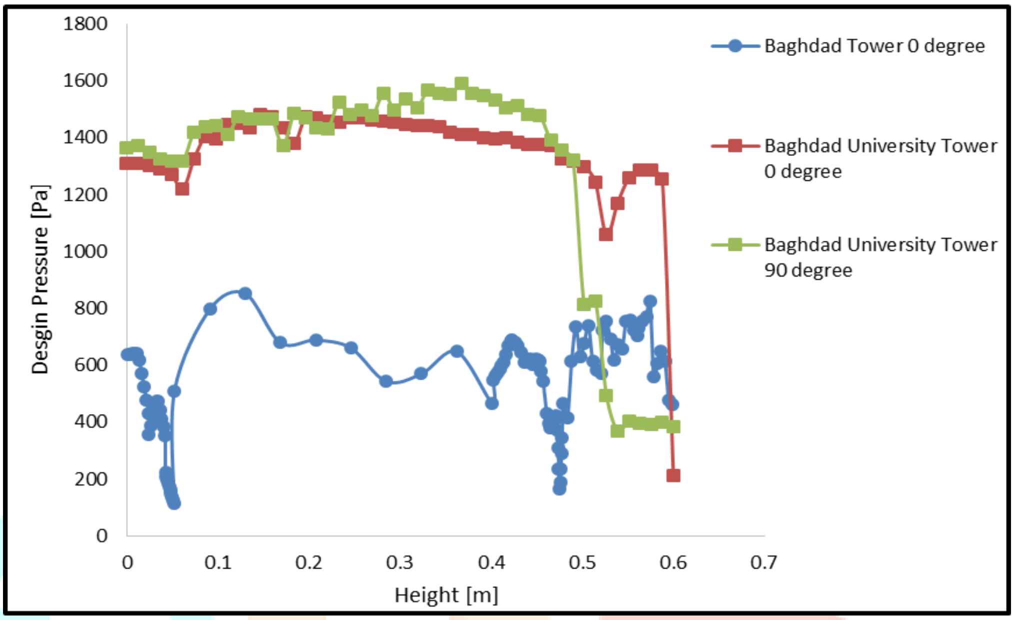


Figure (38a) for 30 m/s Velocity.

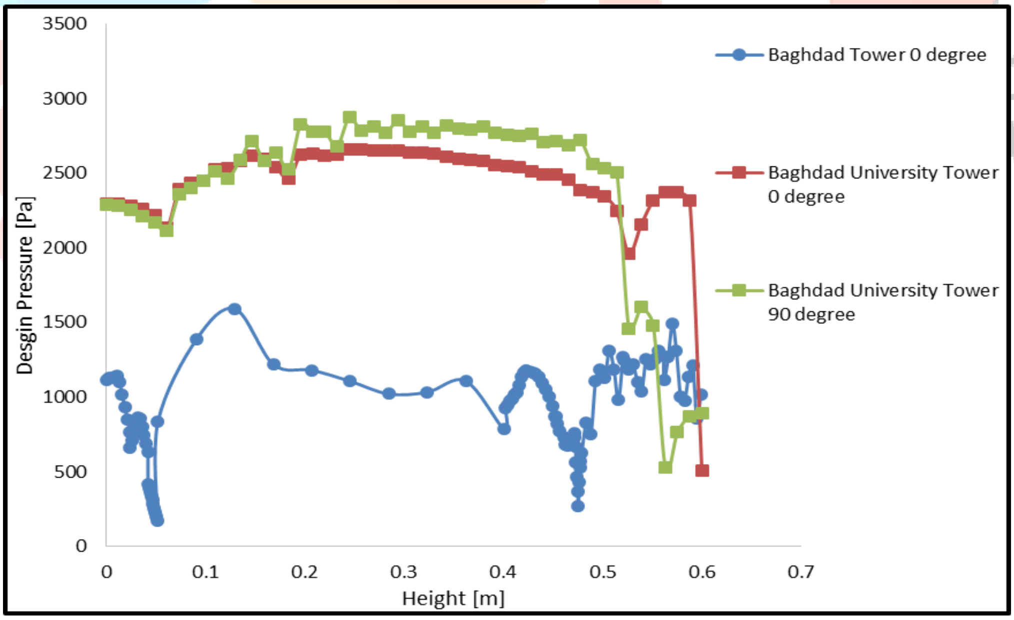


Figure (38b) for 40 m/s Velocity.

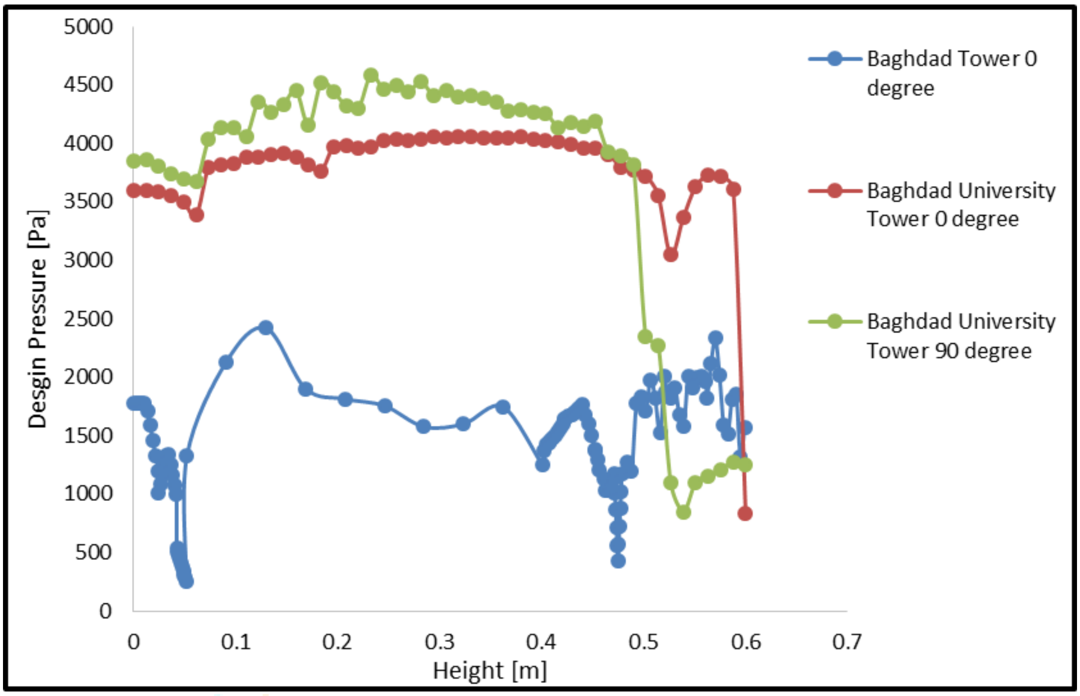


Figure (38c) for 50 m/s Velocity.

Figure (38) Comparison between Baghdad Tower and Baghadad University Tower Models with Different Wind Velocity.

From the results for pressure distribution Contour for both models and with different speeds and wind angles, it can be seen that the pressure on the Baghdad tower is less than the pressure on the Baghdad University tower with the same wind speed. The reason for this difference is due to the shape of the building and the way of it is designed, as the shape of the Baghdad cylindrical tower gives the tower efficiency in resisting the wind. Contrary to the shape of the University of Baghdad, which makes the effect of the wind on it was high. In addition, the effect of the wind on the Baghdad tower with different angles is the same, as its cylindrical shape gives a fixed effect where the wind angles have not changed. While the tower of Baghdad university is affected by the change of wind angles. The results showed that the pressure on the Baghdad University tower at 90 degrees is greater than the pressure at 0 degrees, and this is due to the effect of the surface area on the sides of the building. When comparing the pressure on the University of Baghdad tower at 45 degrees, it can be observed that it is greater than the results of pressure for the rest different angles.

It can be seen from (**Figure 24**) and (**37**) of velocity distribution for two models that the speed on the practical side is the same on the theoretical side and starts to change when exposed to friction. It can be indicated that the wind distribution on the Baghdad tower model shows a slight change in speed as the shape of the cylindrical tower reduces friction and makes the change in speed very little. This is an advantage that the Baghdad Tower has over the rest of the high buildings in Iraq. Upon observing the distribution of speed on the Baghdad University tower, we find that the speed changes clearly and begins with the formation of swirls behind the building and these dummies lead to the production of high clouds pressure that greatly affects the building. It can be concluded that the Baghdad tower is better designed as a building form when looking at the effect of the wind.

Conclusion

In this research paper, the pressure of the model, Baghdad Tower and Baghdad University Tower, was compared between the wind tunnel and CFD results. It can be concluded that the use of CFD instead of examining the costly wind tunnels test. The CFD also provides better outputs than the wind tunnel method. The unlimited use of CFDs can be used for any form, no matter how complex, unlike wind tunnel scanning. In the future, it is possible to use the CFD method in designing buildings that are wind-resistant and to find the most appropriate form of model in terms of economics as well. Researchers should develop this method and code to analyze the structures of high winds and their design. In addition, due to computer evolution, this method can evolve in the future.

References

[1]- ANSYS. Inc. (2013). ANSYS FLUENT Theory Guide. Release 18.2, 15317(November), 373–464.
[2]-Dagnew, A. K., Bitsuamalk, G. T., & Merrick, R. (2009). Computational evaluation of wind pressures on tall buildings. Americas Conference on Wind Engineering.
[3]-K. Wijesooriya and D. Mohotii. (2017). A validated Numerical Approach in Wind Design of Supper Tall Buildings. (August).
[4]-Mahmoud M. Abdelrahman et al. (2016). The Role of CFD Simulation Software in Improving Residential Buildings ' Efficiency : Case Study on Youth Housing in New Damietta. 1–21.
[5]-Mendis, P., Mohotti, D., & Ngo, T. (2014). Wind Design of Tall Buildings , Problems , Mistakes and Solutions. 1st International Conference on Infrastructure Failures and Consequences, (September 2015).
[6]-Paterson and Colin Apelt. (1986). "Computation of wind flows over three-dimensional buildings." Journal of Wind Engineering and Industrial Aerodynamics 24(3), 193–213.
[7]-Samali, B., Kwok, K. C. S., Wood, G.S., Yang, J. N. (2004). "Wind Tunnel Tests for Wind-Excited Benchmark Building." J. Eng. Mech.,. 130, 447-450.
[8]-Simiu, E., Scanlan, R. H. (1978). Wind Effects on Structures, John Wiley & Sons, New York, N.Y.
[9]-Stathopoulos and Zhou. (1995). "Numerical evaluation of wind pressures on flat roofs with the k-ε model." Building and Environment, Volume 30(Issue 2), Pages 267-276.
[10]-Yakhot, V., Orszag, S. A., Thangam, S., Gatski, T. B., & Speziale, C. G. (1992). Development of turbulence models for shear flows by a double expansion technique. Physics of Fluids A, 4(7), 1510–1520.
[11]-W. S. Karrar et al. Comparing of Wind Pressure of a High-Rise Building in Baghdad City between Wind Tunnel and CFD Results. Vol. 05, Issue 01, January 2020.
[12]-W. S. Karrar et al. Comparison of Pressure Results between Wind Tunnel and CFD Simulation for University Baghdad Tower Building. Vol. 05, Issue 03, March 2020.
[13]- W. S. Karrar et al. Wind Pressures Comparison between Wind tunnel and CFD on Square and Triangle Shapes Tall Buildings. Special Issue-2020 15th June.