International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Index Copernicus Value (2015): 78.96 | Impact Factor (2015): 6.391

Validation of Computational Fluid Dynamics Technique for Turbulent Wind Flow Approach, Bluff Two-Dimensional Body

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

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

Abstract: ***This paper studies the suitability of the CFD simulation to study the aerodynamic characteristics of bluff bodies. The bluff body aerodynamics is important to make improvement in understanding wind engineering. During the numerical simulation, the LES turbulence model and CFD Simulations were conducted by Autodesk Flow Design for associated with experimental results. Recent development of CFD technique makes it possible to predict the wind flows around a structure and a structure very close to the actual state. Thus, the practice of CFD even for wind-resistant strategy becomes practically realized now. The assessment of computational wind engineering (CWE) based on computational fluid dynamics (CFD) technique are making the numerical evaluation of wind loads on structures a potentially attractive hypothesis. This paper highlights the fundamental importance of validating CFD simulation from a configuration and a type of flow as close as possible to the topic of our studies. The computed drag coefficient, C_D is estimated by evaluation with the experimental data.***

Keywords: Reynolds number, Wind Engineering, Drag coefficient, LES, CWE

1. Introduction

Wind is the term used for air in motion and is usually applied to the natural horizontal motion of the atmosphere. Motion in a vertical or nearly vertical direction is called a **current**. Movement of air near the surface of the earth is a three-dimensional, with horizontal motion much greater than a vertical motion. Vertical air motion is important in meteorology but is of less importance near the ground surface. On the other hand, horizontal motion of the air, particularly the gradual retardation of wind speed and the high turbulence that occurs near the ground surface, are of importance in building engineering. In urban areas, this zone of turbulence extends to a height of approximately one-quarter of a mile aboveground, and is called the **surface boundary layer**. Above this layer, the horizontal airflow is no longer influenced by the ground effect. The wind speed at this height is called the **gradient wind speed**. In this boundary layer, most of human activities are conducted [1].

In estimation, the wind-induced forces, it is important to diagnose between the aerodynamic characteristics of bluff bodies that are significantly differ from those of the streamlined bodies. Therefore, understanding of bluff body aerodynamics is important to make development in understanding wind engineering.

Computational Fluid Dynamics (CFD) is basically a numerical method to simulating or predicting phenomena and quantities of a fluid flow by solving the equations of motion of the fluid at a discrete set of points. CFD has significantly improved recently. CFD has become an application tool for wind engineering problems [2]. CFD is in general a numericalt echnique in which equations describing the fluid flow are solved on a super-computer. In wind engineering, the airflow is normally the **Atmospheric Boundary Layer (ABL)** flow. Turbulent flows are defined by the well known continuity, momentum and energy equations, named after Navier **and Stokes equations**[3]**.**

Applications of CFD in wind engineering, called computational wind engineering (CWE), have significantly increased in the last two decades. Despite its widespread use, the general evaluation of the approach for quantitative, and sometimes even qualitative predictions, is expressed as lack of confidence. The main objections against practical application of the CWE are the many physical and numerical parameters in the approach, which should be chosen by the user without precise criteria [3].

Therefore, CFD models should be validated through comparison of their results with those of physical models. Based on these validation studies, one can conclude which physical aspects and which numerical parameters should be adopted in the CFD model for accurate simulation of the flow phenomena.

In this study, results of CFD models for circular and rectangular prisms have been compared with the corresponding experimental results for a wide range of **Reynolds number, Re.** The prisms have been assumed so long such that the flow normal to their axes can be considered as two-dimensional flow.

Drag coefficient, C_D, is used to compare between the numerical results and corresponding experimental results. Using C_D gives an indication on the global matching between the numerical simulation and the experimental works. In addition, working in terms of C_D permits to compare the results of numerical simulation with the empirical formulas adopted by the design specifications [4].

Computational fluid dynamics, CFD, simulations for two dimensional shapes not only have academic goals but also have important practical applications specially if one notes that most of design specifications, e.g. (ASCE 7, 2010), use two-dimensional models to distribute wind forces on different faces of the structures.

The CFD software can simulate buildings and structures in their actual scales. This can be considered as another advantage over wind tunnel tests that usually work with a scale down model.

2. Review of Literature

Literature related to the modeling of drag forces of two dimensional bodies that immersed in a turbulent wind flow have been reviewed in this article.

A body immersed in a turbulent wind flow experiences flow-induced fluctuating pressures due to (1) flow fluctuations caused by the presence of the body, and (2) turbulent wind speed fluctuations in the oncoming flow[5].

A. The Bernoulli Equation

For an inviscid (viscosity-free) steady flow, the **Bernoulli equation** indicated below relates the velocity, $\mathbf{V}$, and the static pressure, $\mathbf{p}$, along a streamline:

$$\mathbf{p} + \frac{1}{2}\boldsymbol{\rho}\mathbf{V}^2 = \textbf{constant} \qquad \textbf{Eq. 1}$$

where the first and second term in the sum denote the static and the dynamic pressure, respectively and $\boldsymbol{\rho}$ denotes the fluid density[6].

B. Reynolds number

The Reynolds number is important in determining the extent to which boundary layers are **laminar** or **turbulent**. This in turn determines their ability to withstand an adverse pressure gradient, or their sensitivity to separate from the surface of a body. The Reynolds number, Re, is defined as the ratio of the inertia forces to the viscous forces acting on the fluid particles [7].

$$\mathrm{Re} = \frac{\text{inertia force}}{\text{viscous force}} = \frac{\text{mass} \times \text{acceleration}}{\text{shear stress} \times \text{area}} = \frac{\mathrm{Vl\rho}}{\mu} = \frac{\mathrm{Vl}}{\mathrm{v}} \qquad \textbf{Eq. 2}$$

where $\boldsymbol{\mu}$ is dynamic air viscosity and $\boldsymbol{\rho}$ is air density, $\mathbf{v}$is the kinematic viscosity, and $\mathbf{l}$is the characteristic length.

Reynold's number is important in wind engineering when viscose forces are critical. According to (Simiu, 2011), for air flow at usual temperature and atmospheric pressure conditions, the kinematic viscosity $\mathbf{v}$is approximately $15 \times 10^{-6}\mathrm{m}^2/\mathrm{s}$ for air at 21°C, therefore Reynold's number would be:

$$\mathbf{Re = 67000VL} \qquad \textbf{Eq. 3}$$

where $\mathbf{V}$and $\mathbf{L}$are expressed in terms of m/s and m, respectively.

Reynolds number Re may be interpreted as a quantity of the relative importance of advection (inertia) to diffusion (viscous) momentum changes [5]. If the momentum fluxes are in the same direction, then the Re reveals the atmosphere boundary layer characteristics of the fluid flow. If the fluxes are defined such that the diffusion is in the cross-stream direction, then as shown Figure 1belowReconveys the fluid flow regime (e.g. laminar, turbulent, or transitional) [8].

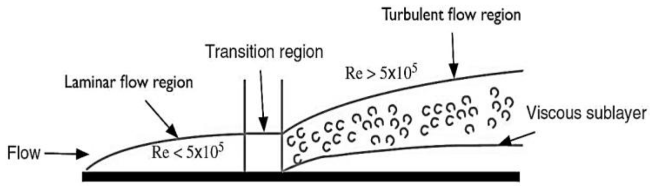


Figure 1: Schematic of the flow over a smooth plate showing the laminar, transitional, and turbulent flow regimes based on the value of Re [8]

C. The Effects of Turbulence on Bluff Body Forces

The turbulence causes early changeover of boundary layers from laminar to turbulent. This usually can have the helpful effect of causing flow separation to be condensed. In the atmosphere, the wind flow is at high Reynolds numbers and one would suppose that all boundary layers on structures are turbulent. Therefore, in the experimental wind tunnel one must have turbulent wind flow to simulate turbulence. However, whether the boundary layers change to turbulent at just the right place will not make too much of an impact on the overall loads, because they are mainly generated through pressure drag, i.e. differences in pressures between the front and back faces. There are special problems with chimneys due to the variation of the separation point with Reynolds number[2].

D. Drag Coefficient $\boldsymbol{A}$.

The drag is the force component parallel to the relative approach velocity exerted on the body by the moving fluid. Drag coefficient, $\mathbf{C_D}$, is a non-dimensional coefficient that usually used in literature to relate drag force resultant to the mean dynamic pressure, $\frac{1}{2}\boldsymbol{\rho V^2}$, of the upstream flow [9].

$$\boldsymbol{C_D} = \frac{\textbf{\textit{Drag force}}}{\boldsymbol{AB}\frac{1}{2}\boldsymbol{\rho V^2}} \qquad \textbf{Eq. 4}$$

where $\boldsymbol{AB}$is typical reference dimensions of the structure.

In general, drag coefficient is a function of the geometry and roughness of the immersed body. It also depends on Reynold's number, $\boldsymbol{Re}$, in the range of laminar and transition zones [7]. The drag coefficient is estimated either based on experimental wind tunnel approach or numerical CFD approach [9].

1. Outlines of CFD simulation

CFD is a technique that developed recently to solve the governing differential equations of fluid dynamics in numerical form [10].

E. Boundary Conditions

In contrast with wind tunnel simulation where boundary layers are included physically, artificially boundary conditions should be imposed in CFD model as the computational domain is finite. Inappropriate boundary layer leads to unreliable results and may lead to breakdown the numerical process [2]. The simulation volume should be selected as large as possible to counteract the effects of artificial boundaries at which the volume has been isolated from the continuum media [11]. Regarding to virtual wind tunnel models, the tunnel dimensions are set large enough compared to the model to avoid edge effects on wind-structure interaction.

F. Turbulence Model

In civil engineering applications, the subsonic incompressible turbulent wind flow is the most common phenomenon. Compressible flow is needed for shocks and high temperature effects [6].

The oncoming flow to the structures is always turbulent since they are immersed in a turbulent atmospheric boundary layer, and the Reynolds number of the structure is very high. The flow phenomena of vortex establishment, interaction between oncoming turbulence flow and separated shear layer and dynamic wake formation are turbulent conditions. The turbulence energy generated at low frequency transfers to the higher frequency region gradually, i.e., first to the energy preservation region and then to the inertia region. The high frequency fluctuation needs to be considered in the simulation because its role in the energy cascade down process cannot be disregarded [2].

Turbulence model is used to dissipate the turbulent energy at high frequency by means of turbulence viscosity, additionally to molecular viscosity. In general, there are two types of turbulence models:

1) Reynolds Averaged Navier–Stokes (RANS)

RANS solve the fluid flow equation (Navier-Stokes equation) by estimating the turbulence effect (turbulence viscosity) through solving supplementary equations of turbulence statistics. The outcomes obtained are averaged over time to reflect the turbulence effect correctly, very complex forms sometimes need to be used. RANS are efficient method if only the mean values of the fluid flow quantities are vital [2].

The major problem is that a direct solution of the Navier-Stokes-equations is only possible for Reynolds-numbers up to approximately 20000, whereas practical applications of fluid flow are in the range of several millions. Thus, turbulence should be modelled by RANS (Reynolds-Average-Navier-Stokes) models allowing for very large Reynolds numbers but do not model all effects, while LES (Large Eddy Simulation) has gained some popularity, nevertheless it requires high computational effort [2].

2) Large Eddy Simulation, LES

LES solves the large scale (grid scale) components of the fluid flow directly with the small scale (sub-grid) components modeled. This enables time dependent calculation. LES is very time-consuming since it generally conducts three-dimensional calculations, and long time series data of the fluid flow quantities are required to achieve reliable analysis. The formulation of LES is simpler than that of RANS, its dependence on the flow phenomenon with few parameters. With LES, the time series data of the flow quantities at all grid points are obtained. LES is allowing evaluation of the mean and peak values as well as the power spectra of the fluctuation at all points. Thus, LES is more beneficial than RANS for wind engineering. However, the filter operation of LES is averaging process over space. It is necessary to note that the peak value obtained in LES has been subject to a filter process[2]. Theoretically, the LES simulations are more desirable than the RANS for the study of wind around structures since they solve precisely the large eddy and simulate only the smallest eddies [11].

3. Validation Process

CFD simulations results may be validated from wind tunnel tests, which give more accurate results for the same configuration that measured on a real site and are also based on the assumption of a static boundary conditions. Wind tunnel is a technology used with great success for decades to assess precisely the problems of wind around buildings.

One can carried out a validation of Autodesk Flow Design software as a tool for simulation of wind around buildings by comparing the simulation results with those of wind tunnel tests available in the literature. In this study the validation was executed for the following two dimensional four cases:

- ➢ Validation of 2D (Circular Prism),
- ➢ Validation of 2D (SquarePrism),
- ➢ Validation of 2D Rectangular Prism with Effect of Corner Radius,
- ➢ Validation of 2D Rectangular Prism with Effect of Narrowness.

G. Validation of 2D (Circular Prism)
1) Experimental Results

Drag coefficient of circular prisms for a wide range of Reynolds numbers is presented in Figure 2. For a smooth cylinder, the drag coefficient is about 1.2 for the Reynolds number range from 10^4 to 2×10^5. Then, it shows a rather sudden drop to 0.3 at a Reynolds number of 5×10^5 after which it gradually increases again [2]

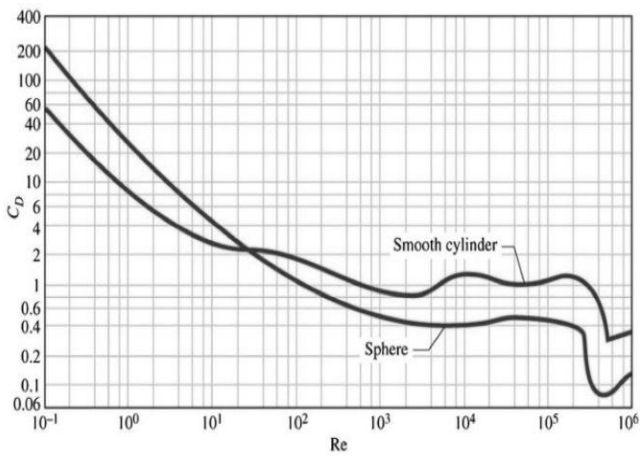


Figure 2: Drag coefficient for smooth spheres and circular cylinders as a function of Reynolds number [2]

This rather unusual behavior of the drag coefficient can be explained by the behavior of the boundary layer on the surface of the cylinder, which results in the flow regimes shown schematically in Figure 3below. At low Reynolds numbers the boundary layer is laminar and is very prone to the adverse pressure gradient on the rear of the cylinder, and separation occurs at θ equal to 82^oproducing a large negative pressure wake and a drag coefficient of about 1.2. At higher Reynolds numbers the boundary layer becomes turbulent before this location and is better able to withstand the adverse pressure gradient, and so it does not separate until θ equal to 120^o, resulting in a much smaller wake region, a higher pressure on the rear (base), and a reduction in the drag coefficient to 0.3[2].

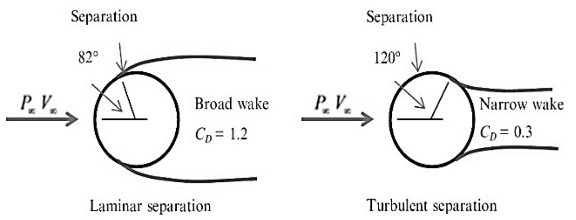


Figure 3: Flow past a circular cylinder showing laminar and turbulent separation [2]

According to Emil Simiu & Scanlan, Robert H, (1996) for a prism of circular cross section in smooth flow the variation of mean drag coefficient C_D is dependent on Reynolds number as indicated in Figure 4. Note particularly how $C_D$drops sharply in the range about $2 \times 10^5 \leq Re \leq 5 \times 10^5$. This region of sharp drops is called a critical region and corresponds to a condition wherein the transition from laminar to turbulent flow occurs in the boundary layer that forms on the surface of cylinder. The turbulent mixing that takes place in the boundary layer helps transport fluid with higher momentum towards the surface of the cylinder. Separation then occurs much farther back and the wake consequently narrows, finally producing a value of the time-averaged C_D that is only about $^1/_3$ of its highest value. As Re increases into supercritical and then transcritical range $(Re \geq 4 \times 10^6)C_D$ increases once more but remain much lower than its supercritical values [6].

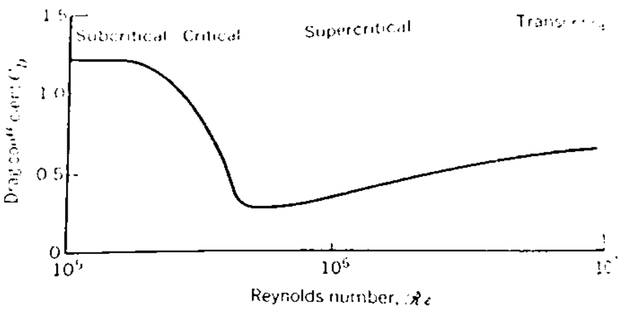


Figure 4: Evaluation of mean drag coefficient with Reynolds number for a circular cylinder [6]

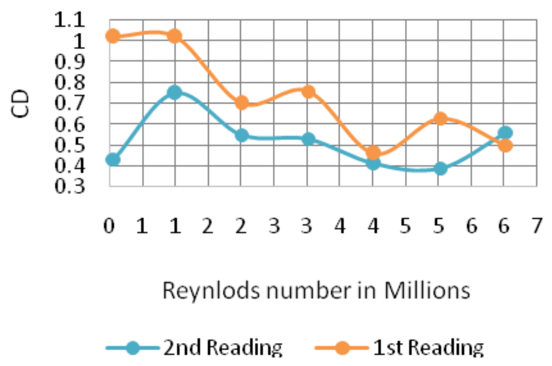
According to (Stathopoulos, Ted & Baniotopoulos, Charalambos C, 2007), the flow around the cylinder is depends on the Reynolds number. The different ranges of Re are described as:

- Subcritical region approximately in the range $300 < Re <$: The boundary-layer flow at the cylinder surface is laminar, and the points of separation are about from the point of stagnation. Vortices are shed from alternate sides of the cylinder within a relatively narrow frequency band. Thus, the auto spectrum of the lateral load has a characteristic maximum at the predominant frequency of vortex shedding.
- Supercritical region approximately in the range. $10^5 < Re < 3.5 \times$: The points of separation are on the leeward and the wake has become much narrower. The range gives the transition to the transcritical range and the lateral load is a periodic and random. The auto spectrum has no characteristic.
- Transcritical region approximately in the range $3.5 \times 10^6 <$: The boundary layer flow at the cylinder surface is turbulent, and the points of separation are about ː from the point of stagnation and the wake is narrower than in the subcritical range. However, it is wider than in the supercritical range. Finally, vortices are shed from alternate sides of the structure within a relatively narrow frequency band, therefore resembling the subcritical range [12].

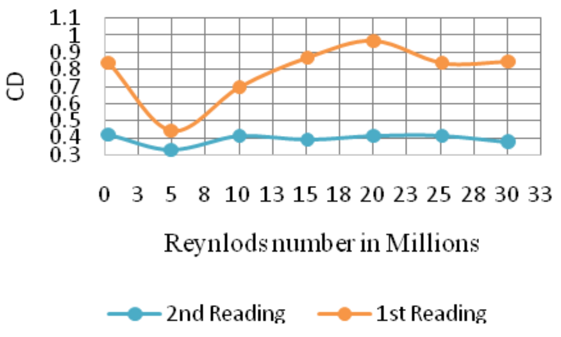
Albertson, Barton, & Simons, 1960 state that, at Re in the range 2×10^5 for both sphere and the cylinder, the drag coefficient suddenly decreases to about one-third its original magnitude. This phenomenon is caused by a change in nature of the boundary layer from a laminar boundary layer to a turbulent boundary layer. For a circular cylinder with axis perpendicular to the flow, at Re in the range of 10^5 the drag coefficient is 1.2 and for high Re (more than 5×10^5) the drag coefficient is about 0.33[13].

2) Results Extracted from Software

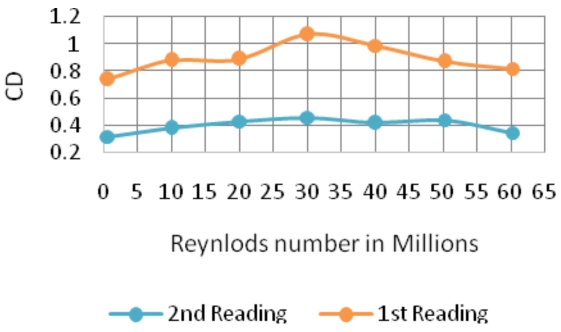
During this study, a maximum wind of $90\ m/s$ is adopted with an increment of $15\ m\ /s$. Different diameters including 1.0m, 5.0m, and 10m have been considered to cover all domain of Reynolds number. Results for drag coefficient, C_D, determined with Flow Design is presented in Figure 5.



(a)



(b)



(c)

Figure 5: Drag coefficient for circular cylinders as a function of Reynolds number (a) Radius 500mm (b) Radius 2500mm (c) Radius 5000mm

H. Validation of 2D (Square prism)

1) Experimental Results

(Emil Simiu & Scanlan, Robert H, 1996) state that in some cases the effect of turbulence on the drag force can be important. For members with square cross section. This effect depends upon (1) the ratio b/hbetween the sides of the cross section and (2) the turbulence in the oncoming flow. If the ratio b/h is small. no now reattachment occurs following separation at the front comers. For two-dimensional square cylinder the drag coefficient is 2.03[6].

According to (Streeter, Victor L & Wylie, E Benjamin, 1985) the drag coefficient for square tube is a function of Reynolds number as shown below:

- equal to 2 for equal to $3.5 \times$ with an angle of 0°.
- equal to 1.6 for equal to 10^4 to with an angle of 45°.

Tamura, Yukio & Kareem, Ahsan, (2013) state that, for a rectangular prism immersed in a uniform stream of fluid. In this condition, it is evident that although the onset flow is smooth and without turbulence, the body produces a large wake, which is unsteady and fluctuates from side to side. Although the Bernoulli equation can be used in the flow far from the wake to connect the local pressures and velocities to the freestream values, this is not the case in the wake, where the sum of the static and dynamic pressures is less than the freestream values. This loss in mechanical energy of the fluid has been converted to turbulence. However, real flows are more complicated than the situation shown in Figure 6. Wind flows are turbulent and the mean speed increases with height above the ground [2].

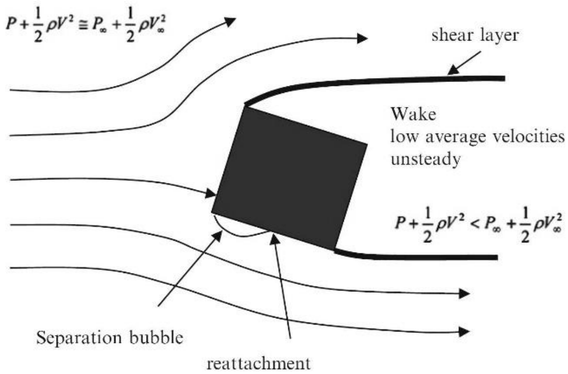


Figure 6: Bluff body in smooth uniform flow [2]

The free-stream turbulence controls the pressure instabilities on the upstream face of a building. It also determines the shear layer performance, as shown in Figure 7. The shear layer performance is crucial for the flow around the building and so for the mean and instable pressures acting on the building. Increasing the free-stream turbulence increases the rate of entrainment of wake fluid into the more turbulent shear layers. Which in turn fluctuations the pressures acting on the building [14].

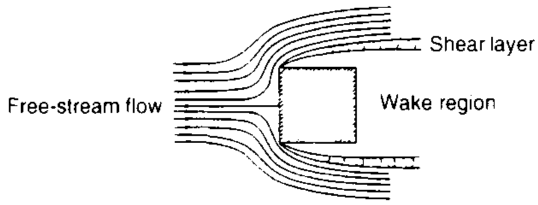


Figure 7: Time-average pattern of the flow past a rectangular tube[14]

According to (Boye Ahlborn, Mae L. Seto, & Bernd R. Noack, 2002) the drag coefficient for square tube is a function of Reynolds number as shown in Table 1.

Table 1: Experimental data and derived quantities for square tube

Source	Re	C_D
Square cylinder	1.76×10^5	2.04
	10^5	2.05
	13000	0.216

2) Results Extracted from Software

During this study, a maximum wind of $90\ m/s$ is adopted with an increment of $5\ m\ /s$ ($1, 5, 10,$ and $15\ m/s$) to cover all the domain of turbulence effect in the critical region and for speed of ($30, 60,$ and $90\ m/s$) to cover all domain of Reynolds number. The width that adopted is $1\ m$.Results of drag coefficient, C_D, determined with Flow Design is presented in Figure 8 for square cylinder with an angle 0° and Figure 9 for an angle 45°.

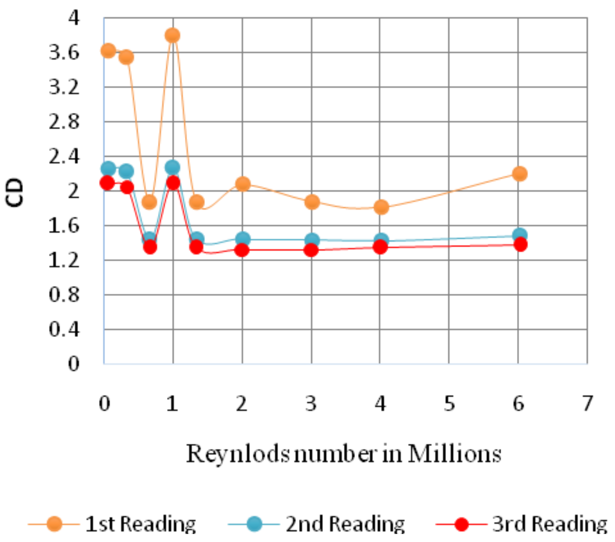


Figure 8: Effect of Reynolds number on drag coefficient (Angle 0°)

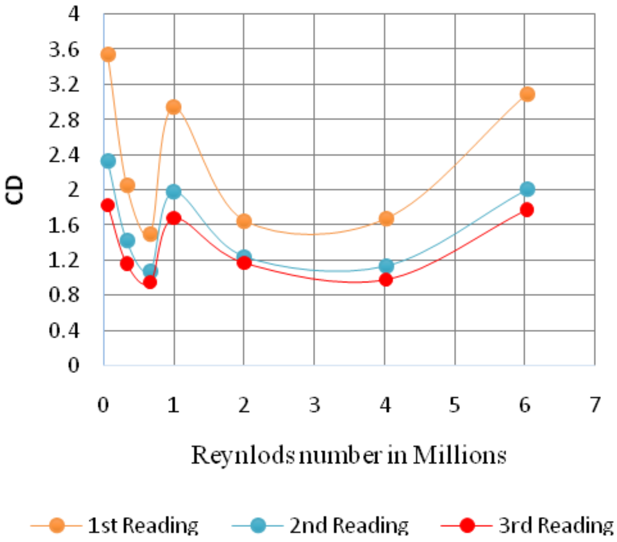


Figure 9: Effect of Reynolds number on drag coefficient (Angle 45°)

I. Validation of 2D Rectangular Cylinder "Effect of Corner Radius"

1) Experimental Results

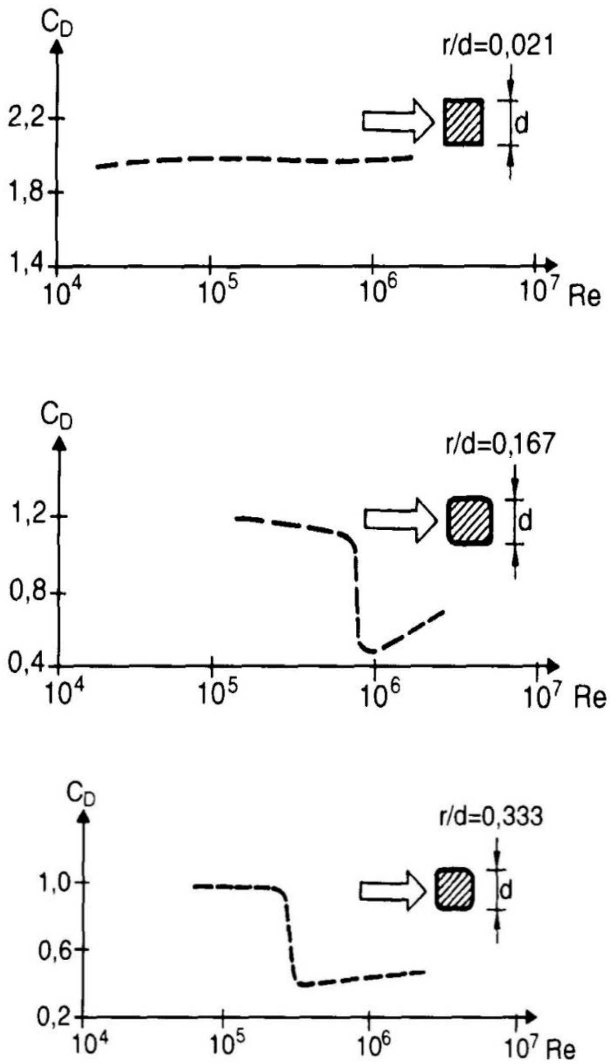
According to Emil Simiu & Scanlan, Robert H, (1996) Figure 10belowexplains the evaluation with Reynolds number of the mean drag coefficient of a square in smooth flow during succeeding modification of its corners. Note that only the sharp cornered square shows practically unchanging drag with Reynolds number. This is basically accounted for by the initial separation of the flow at upstream corners and shortness of the after body that essentially prevents the opportunity of flow reattachment, while squares with rounded corners tend to hold the same kind of the critical region for the drag coefficient as realized for a circular cylinder [6].

The flow around bodies with sharp ends is practically independent of Reynolds number, as shown in Figure 10, landscape roughness is the totally governing restriction. The demand for correct scaling of landscape roughness was first expressed by Martin Jensen, and nowadays is called Martin Jensen`s model law [14].

For square tubes or structures with sharp edges on their outline, C_D; is practically independent of R_e. This can be obviously observed in Figure 10which indicates the difference of drag coefficients for regularly increasing the radius of curvature of structure corners as we go from an almost square to a fully circular shape [12].

Tamura, Yukio & Kareem, Ahsan, (2013) state that the corner radius on bluff body shapes is very significant, as it can support or suspension the way in which the flow goes past a corner. Sharp corners encourage separation, and the flow will separate there no matter what the is R_e. This is the reason that boundary layer wind tunnels can be used for the majority of wind engineering investigations at Reynolds numbers that are low by three orders of magnitude.

The Figure shows that sharp corners are insensitive to Reynolds number. As the corner radius is increased, it can be realized that the drag reduces when the R_e. is increased above a certain value, and that this value gets lower with increased radius [2].



International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Index Copernicus Value (2015): 78.96 | Impact Factor (2015): 6.391

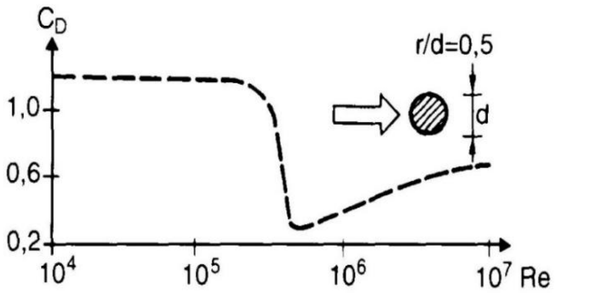
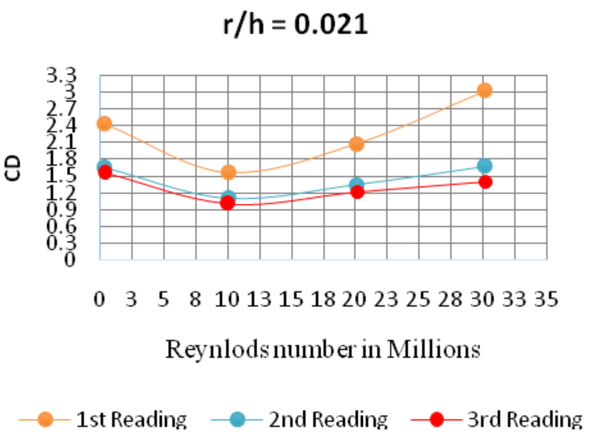
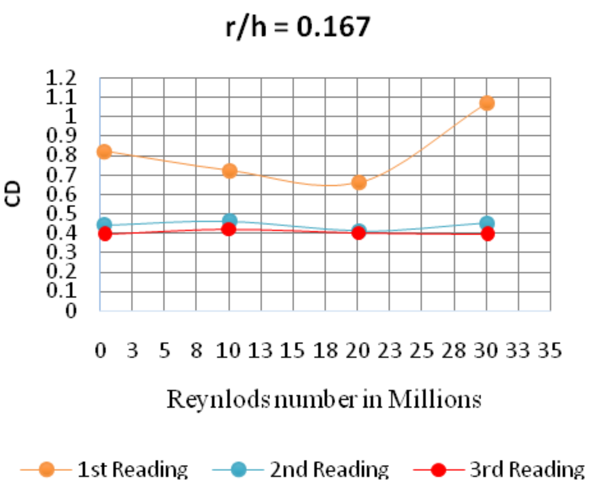


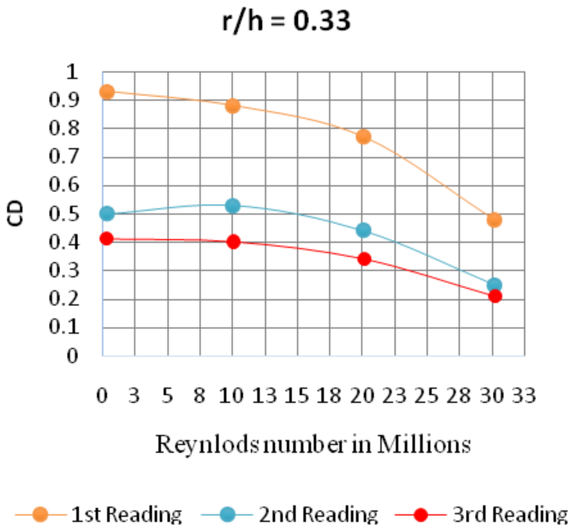
Figure 10: Dependence of the drag coefficient C_D upon R_e. and the radius r of curvature at the edges (Scruton and Rogers, 1971) ([12], [6])

2) Results Extracted from Software

During this study, a maximum wind of $90\ m/s$ is adopted with an increment of $30\ m\ /s$ $(1, 30, 60, \text{and}\ 90\ m/s)$ to cover all the domain of turbulence effect. Different corner radius is taken to cover all the early separation of the flow at the upstream corners of the body. Results for drag coefficient, C_D, determined with Flow Design is presented in Figure 11.







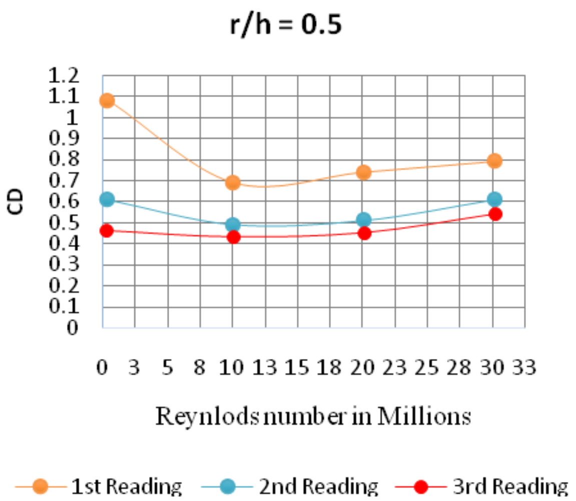


Figure 11: Effect of Reynolds number and corner radius on drag coefficient, square to rounded cylinder (r is the corner radius; h is width of cylinder perpendicular to the flow)

J. Validation of 2D Rectangular Cylinder "Effect of Narrowness

1) Experimental Results

The drag coefficient of elongated rectangular-section body in smooth flow is a function of the narrowness of its wake, nevertheless the lower bound of wake width is approximately the full width of the body. The wake width at somewhat lower C_D; then, when flow reattachment to the body starts to happen, the drag coefficient C_D drops. This is a purpose primarily of the elongation b/h of the body, as shown in the Figure 12. Flow in the critical region is accompanied by turbulence, and therefore the region between high R_e and low R_e band of possible values [6].

The stream-wise length of the bluff body acting a significant part on the C_D, as shown in Figure 12. Again, flow characteristics such as the position of re-attachment, if it exists at all, are determinants of the magnitude of pressure-induced drag force. Obviously, the size of wake will effect on the drag force; therefore, for two structures with the identical frontal area, that with the longer length, i.e. the narrower wake, will experience the lesser drag force. Turbulence of the oncoming flow causes re-attachment to occur at a fairly smaller length, therefore turbulence also affects drag forces. This reveals the effect of the upstream exposure to the wind-induced pressures on structures [12].

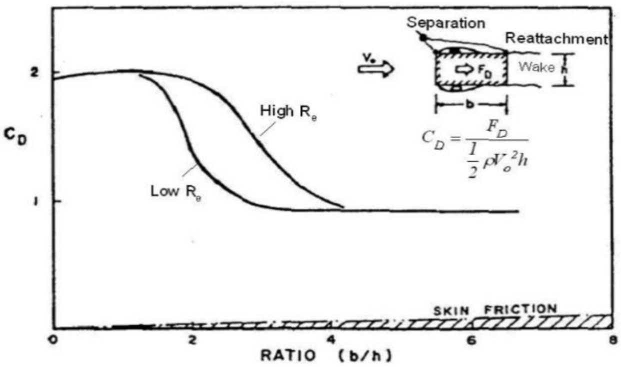
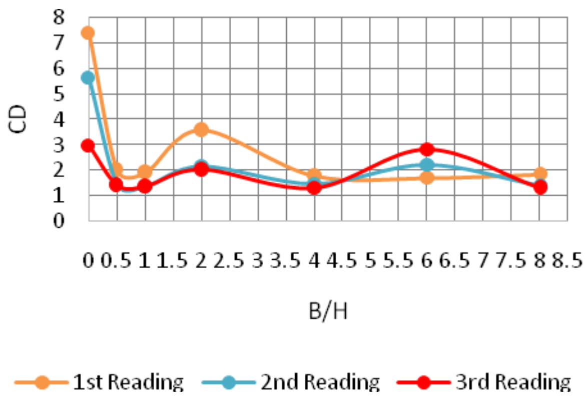


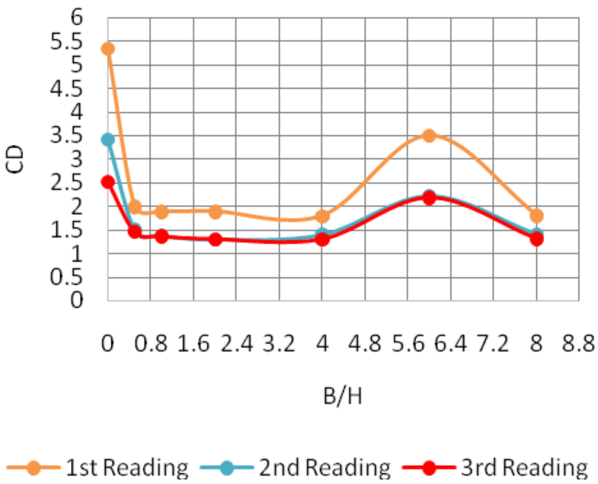
Figure 12: Effect of after-body upon drag of a rectangular cylinder [12]

2) Results Extracted from Software

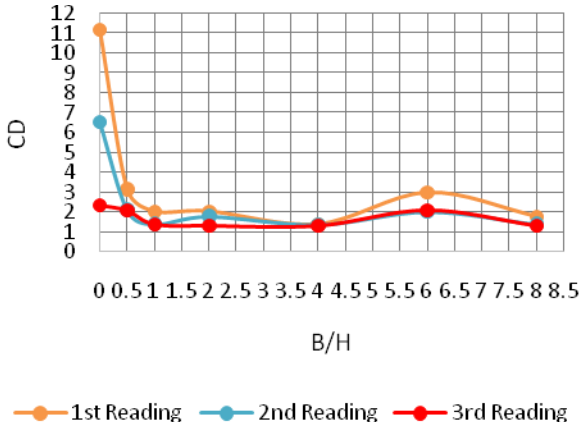
During this study, a maximum wind of $90\ \mathrm{m/s}$ is adopted with an increment of $45\ \mathrm{m/s}$ $(1, 45, 90\ \mathrm{m/s})$ to cover all the domain of turbulence effect. Different aspect ratio including 0, 1, 2, 4, 6, and 8 to cover all the domain of narrowness of the body. Results for drag coefficient, C_D, determined with Flow Design is presented in



(a)



(b)



(c)

Figure 13: Relation between drag coefficient with different aspect ratios (a) velocity=1m/s (b) velocity=45m/s (c) velocity 90m/s

4. Conclusions

The numerical methods presented intended for the solution to the flow problem, and numerical simulation of the air flows, together with the parametric studies, will permit engineers and designers to get a good representation of the distribution of the wind pressure around bodies. The results of cross-comparisons for the four test cases conducted inside this development will be utilized to validate the accuracy of CFD used in the real-world applications of wind calculations.

The accuracy of results depends upon boundary conditions, meshing, and turbulence of the model and defining the physical property values precisely as the realistic conditions.

The drag coefficients, streamline, velocity vector, and numbers of associated variables. Through the model surface can be determined with the benefit of CFD analysis.

The practice of CFD permits the engineer to utilize the techniques developed by designers to study the accurateness of RANS and LES when combined to structure aerodynamics, or bluff bodies, which is distinguished by flow separation at structure corners and high Reynolds numbers (turbulent flow). CFD provides continuous visualization of aerodynamic information on the entire surface areas of the structure instead of the discrete point measurements extracted from wind tunnel study. Additionally, CFD can provide detailed flow visualizations used for the fluid field around the structure which information on flow separation, recirculation zone reattachment, and vortex shedding can be extracted.

References

[1] B. Taranath, Wind and earthquake resistant buildings: structural analysis and design, John A. Martin & Associates, Inc., 2004.

[2] Tamura, Yukio and Kareem, Ahsan, Advanced structural wind engineering, Springer Science & Business Media, 2013.

[3] J. Franke, C. Hirsch, A. Jensen, H. Krüs, P. W. &. S. Miles, J. Wisse and N. Wright, "Recommendations on

the use of CFD in wind engineering, " 2004.

[4] ASCE 7, Minimum Design Loads for Buildings and Other Structures, ASCE/SEI, 2010.

[5] E. Simiu, design of buildings for wind a guide for ASCE 7-10, Johm Wiley & Sons, Inc., 2011.

[6] Emil Simiu and Scanlan, Robert H, Wind Effects on Structures: Fundamentals and Application to Design, New York: John Wiley & Sons, 1996.

[7] Streeter, Victor L and Wylie, E Benjamin, Fluid Mechanics, McGraw-Hill, 1997.

[8] F. Moukalled, L. Mangani and M. Darwish, The Finite Volume Method in Computational Fluid Dynamics, Springer International Publishing, 2016.

[9] Shames, Irving Herman, Mechanics of fluids, McGraw-Hill New York, 1982.

[10] J. D. Anderson and J. Wendt, Computational fluid dynamics, Springer, 1995.

[11] S. Reiter, "Validation Process for CFD Simulations of Wind Around Buildings, " 2008.

[12] T. Stathopoulos and C. Baniotopoulos, "Wind Effect on Buildings and Design of wind-sensitive structures, " Springer Science & Business Media, 2007.

[13] M. L. Albertson, J. R. Barton and D. B. Simons, Fluid mechanincs for Engineers, Prentice- Hall, Inc., 1960.

[14] Claës Dyrbye and Svend Ole Hansen, Wind Loads on Structures, John Wiley & Sons, Ltd., 1997.

[15] B. S. Taranath, Structural analysis and design of tall buildings: steel and composite construction, CRC press, 2012, pp. 212-214