A 3D Numerical Investigation into the Effect of Rounded Corner Radii on the Wind Loading of a Square Cylinder Subjected to Supercritical Flow

Nivedan Vishwanath, Aditya Karthik Saravanakumar, Kush Dwivedi, K Ram Chandra Murthy, P S Gurugubelli, Sabareesh G R

Department of Mechanical Engineering,
BITS Pilani Hyderabad Campus, Hyderabad, India 500078.

Abstract

Tall buildings are often subjected to steady and unsteady forces due to external wind flows. Measurement and mitigation of these forces becomes critical to structural design in engineering applications. Over the last few decades, many approaches such as modification of the external geometry of structures have been investigated to mitigate wind-induced load. One such proven geometric modifications involved rounding of sharp corners. In this work, we systematically analyze the effects of rounded corner radii on the flow-induced loading for a square cylinder. We perform 3-Dimensional (3D) simulations at a high Reynolds number of $Re = 1 \times 10^5$ which is more likely to be encountered in practical applications. An Improved Delayed Detached Eddy Simulation (IDDES) formulation is used with the $k-\omega$ Shear Stress Transport (SST) model for near-wall modelling. IDDES is capable of capturing flow accurately at high Reynolds numbers and prevents grid-induced separation near the boundary layer. The effects of these corner modifications are analyzed in terms of the resulting mean and fluctuating components of the lift and drag forces compared to a sharp corner case. Plots of the angular distribution of the mean and fluctuating pressure coefficient along the square cylinder’s surface illustrate the effects of corner modifications on the different parts of the cylinder. The windward corner’s separation angle was observed to decrease with an increase in radius, resulting in a narrower and longer recirculation region. Furthermore, with an increase in radius, a reduction in the fluctuating lift, mean drag, and fluctuating
drag coefficients has been observed.

Keywords: Detached eddy simulations (DES), Rounded corner modifications, Wind loading, Supercritical flow, Square cylinder

1. Introduction

Tall structures such as buildings and wind turbines encounter unsteady pressure forces which usually have an undesirable impact on their integrity. It is understood that the origin of these forces lies in the shear layer instability and vortex shedding phenomena in the wake region of these structures. Many different approaches have been explored to mitigate such wind loading over the past few decades, including the manipulation of the oncoming flow, the wake behind the structures, or both. Some of the commonly investigated approaches include corner modifications [1, 2], construction of wind fences around the building [3, 4], and the use of escarpments [5].

Corner modifications delay the boundary layer separation and diminish vortex shedding in the wake, thereby leading to reduced lift and drag forces on the cylinder. A vast range of corner modifications have been studied over the years, including chamfered and rounded corners, slotted corners [6, 7], cuts, and tapers [8, 9, 11]. The effect of tapered corner modifications on the across-wind displacement response was studied in [8]. It was concluded that the taper effect is observed when the range of reduced velocity is high and the structural damping ratio is low. With regards to chamfered corner modifications, an experimental investigation of chamfered square cylinders in the presence and absence of an oscillation frequency was carried out in [10]. More recently, a rectangular building model with varying degrees of chamfer was experimentally investigated in [11]. It was observed that an increase in chamfering led to a decrease in wind loading. A series of experiments presented in [11] concluded that among three types of corner modifications (cut, recession, and roundness), rounded modifications were the most effective in suppressing the aeroelastic instability for a square prism. It was also observed that the wind-induced vibration reduced when the corner radius was increased.

One of the earliest attempts to experimentally study the effects of corner modifications is described in [12]. The results from the study pointed at a sharp jump in the mean drag force of a corner modified cylinder at Re = 7 × 10^5 and r/D = 0.167. Drag-crisis was observed, a phenomenon in which the time-averaged drag gradually decreases with an increase in the Re. The mean drag eventually reaches its point of minima
before increasing again. The Reynolds number regimes before and after this significant decrease in drag are called the subcritical and supercritical regimes, respectively. In the study by Lee [13], the mean and fluctuating fields across a 2D square prism at $Re = 1.76 \times 10^5$ were measured. Okamoto and Uemura [14] experimentally studied the effect of corner radius on the flow around a cube at $Re = 4.74 \times 10^4$, corresponding to the subcritical regime. A decrease in drag coefficient was noted with an increase in the corner radius ratio. An increase in the corner radius for square cylinders lead to increased flow reattachment, as observed in [15]. The results presented in [15] also highlighted the contribution of inflow turbulence towards the flow reattachment on the side walls of a corner modified cylinder in the subcritical flow regime. A comprehensive study on the near wake of a corner modified square prism was done in [16]. It experimentally studied the flow around a square prism for a set of corner radii $r/D = \{0, 0.157, 0.236, 0.5\}$ at $Re = 2600 \& 6000$, using particle imaging velocimetry (PIV). The results indicated that as the $r/D$ value was increased from 0 (corresponding to a sharp square cylinder) to 0.5 (corresponding to a circular cylinder), the strength of vortex shedding decreased and the length of vortex formation doubled. This suggested that the leading edge took precedence over the trailing edge in influencing the near wake structure at low $Re$ [17]. More recently, Carassale focused on the transition from subcritical to the supercritical regime in smooth and turbulent flow conditions and made observations about the effect of rounded corner modifications on the aerodynamic characteristics of the wake [18]. Three cases with $r/D = \{0.067, 0.133\}$ were considered and experiments were conducted for ($1.7 \times 10^4 \leq Re \leq 2.3 \times 10^5$). In addition to the other observations, the effect of different angles of incidence on the aerodynamic performance of the cylinders was also analysed.

While experimental studies give information about the aerodynamic characteristics in the wake of these cylindrical structures, they cannot be used to study the complex flow interactions produced as a result of the corner modifications. Therefore, in addition to experiments, numerical simulations have been performed to understand flow interactions with the cylinder with better clarity. Shi [19] carried out 2D and 3D LES simulations of flow around a rounded corner square cylinder at $Re = 2600$. It was observed that while both 2D LES and 3D LES are capable of predicting the large-scale vortical structures accurately, only 3D LES can capture the forces and recirculation length satisfactorily. A comprehensive analysis on the development of separated and transitional flow around corner-modified cylinders for $Re = 1000$ was done in [20] by performing direct numerical simulations (DNS). The results indicated that the length of the time-averaged recirculation region was highest for a corner radius ratio of $r/D = 0.125$. A study on the
effect of the angle of incidence on the drag characteristics of a rounded corner square cylinder was presented in [21]. The drag coefficient attained minima when the angle of incidence was between 5 and 10 degrees. Cao and Tamura [22] simulated subcritical and supercritical flows around square cylinder with a rounded corner ratio $r/D = 0.167$. Two Reynolds numbers, \( \text{Re} = 2.2 \times 10^4 \) and \( \text{Re} = 1.0 \times 10^6 \) were considered, denoting the subcritical and supercritical regimes, respectively. A decrease in the time-averaged drag value and an increase in Strouhal number was noted for the supercritical case. They further extended their work by studying the shear-inflow effects for the above simulated cases in [23] and [24]. Other numerical studies on rounded corner cylinders are summarised in [25, 26].

In order to observe complex flow interactions in the wake of these cylinders, high fidelity methods like the direct numerical simulation (DNS) and large-eddy simulations (LES) are commonly employed. These methods yield satisfactory results for the flow in the wake, and can also resolve wall interactions with good accuracy [27, 28, 29]. However, these methods require extremely fine grids and their degree of refinement increases with an increase in \( \text{Re} \). This makes them computationally expensive and often unfeasible for high \( \text{Re} \) simulations. The detached eddy simulation (DES) is an intermediate scheme that offers a solution accuracy similar to LES at a modest computational cost. Some recent studies have used the DES method to study flow around corner modified cylinders [30].

Owing to the high computational cost, there are very few studies that deal with flow around bluff bodies at very high Reynolds number. Furthermore, most of them consider very few corner modified configurations for their study. The present study aims to provide a systematic analysis of the effects of rounded corner modifications on the flow characteristics around a square cylinder. A very high Reynolds number of \( \text{Re} = 1 \times 10^5 \) has been considered to study the problem in a practical context. Five cases with increasing normalised corner radius (also referred to as rounding radius) \( r/D = 0, 0.05, 0.10, 0.15, \) and \( 0.20 \) are simulated in this study. Here, \( r \) is the rounding radius of the corners and \( D \) is the diameter of the square cylinder. An improved delayed detached eddy simulation (IDDES) formulation has been used to capture flow separation with high accuracy at a moderate computational cost. The results from the sharp corner square cylinder simulation are validated against published LES simulations, thereby proving the applicability of DES to such problems. A strong emphasis is placed on understanding the effect of different corner radii on the force coefficients and the mean flow behaviour. The work analyses the following things: (1) The trend of force

4
coefficients on the cylinder with varying corner radii; (2) The variation of mean and fluctuating values of pressure coefficient along the surface of the cylinder; (3) The flow separation and recirculation region patterns as a consequence of corner modifications.

In Section 2, we describe the IDDES formulation employed in the current simulations and discuss the problem statement in terms of the computational domain, mesh and boundary conditions. This is followed by Section 3 which presents a discussion on the observations based on the study by comparing the resulting forces of the cylinder and flow patterns for different corner radii. Finally, Section 4 briefly summarizes the findings from this study.

2. Numerical Method

Detached eddy simulation (DES) is a hybrid approach that employs RANS models in the near-wall region and recovers LES outside the boundary layer. Since the instability in the flow often dominates over a small part of the computational domain, this method yields reasonably accurate results at high Reynolds numbers at a modest computational cost. However, the standard DES model comes with drawbacks such as the modelled stress depletion (MSD) [31] and grid induced separation (GIS), which may occur due to inadequate mesh refinement near the walls or a growing boundary layer. The delayed DES (DDES) [32] tackles these issues utilizing a modified DES length scale to ensure that the RANS model is used throughout the boundary layer. More recently, the improved delayed DES (IDDES) was proposed as a combination of DDES and wall-modelled LES (WM-LES). It further helps with preventing GIS by increasing the modelled stress contribution across the interface. This IDDES formulation, along with the $k - \omega$ SST RANS model [33] has been used in this work.

2.1. Governing Equations

The Reynolds-Averaged Navier-Stokes equations, which govern the turbulent flow of incompressible, viscous fluids, are given as

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0,$$

(1)

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_l}{\partial x_l} \right) \right] + \frac{\partial}{\partial x_j} \left( - \rho u_i u_j \right),$$

(2)
where $u_i$ is the $i^{th}$ component of the fluid mean velocity vector $\bar{u}$, $p$ is the mean pressure, $\rho$ is the density and $\mu$ is the dynamic viscosity of the flow. The Reynolds stress term $\left(-\rho \overline{u_i u_j}\right)$ must be solved in order to close the above equations. The transport equations for the turbulent kinetic energy $k$ and the specific dissipation rate $\omega$ are given as

$$\frac{\partial \rho k}{\partial t} + \nabla \cdot (\rho \bar{u} k) = \nabla \cdot [(\mu + \mu_T \sigma_k) \nabla k] + P_k - \frac{\rho k^{3/2}}{l_{IDDES}}$$

(3)

$$\frac{\partial \rho \omega}{\partial t} + \nabla \cdot (\rho \bar{u} \omega) = \nabla \cdot [(\mu + \mu_T \sigma_\omega) \nabla \omega] + (1 - F_1) 2\rho \sigma_{\omega,2} \frac{\nabla k \cdot \nabla \omega}{\omega} + \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2$$

(4)

$$\mu = \frac{\rho k}{\max(a_1 \cdot \omega, F_2 \cdot S)}$$

(5)

where $\mu_T$ is the turbulent viscosity, $\sigma_k$ and $\sigma_\omega$ are the turbulent Prandtl numbers, $P_k$ is the generation of turbulent kinetic energy due to the mean velocity gradient and $F_1$ and $F_2$ are the blending function. $a_1$ is a model constant and is equal to 0.31, while $S$ denotes the magnitude of the strain rate tensor. $\sigma_{\omega,2}$, $\alpha$ and $\beta$ are constants associated with the cross-diffusion, the generation of $\omega$ and the dissipation of $\omega$, respectively.

The IDDES length scale is defined as functions of the RANS and LES lengths scales, $l_{RANS}$ and $l_{LES}$ as,

$$l_{IDDES} = \tilde{f}_d (1 + f_c) l_{RANS} + (1 - \tilde{f}_d) l_{LES}$$

(6)

$$l_{RANS} = \frac{k^{1/2}}{\beta^* \omega}$$

$$l_{LES} = C_{DES} \Delta$$

(7)

where $C_{DES}$ and $\beta^*$ are empirical constants and $\Delta$ is defined as $\Delta = \min \{\max(C_w d, \Delta_{max}, \Delta_{min})\} \Delta_{max}$

with $d$ as the near-wall distance, $\Delta_{min} = \min(\Delta x, \Delta y, \Delta z)$ and $\Delta_{max} = \max(\Delta x, \Delta y, \Delta z)$.

A detailed formulation of the functions $\tilde{f}_d$ and $f_c$ can be found in [34].

2.2. Computational Model

The present study focuses on the flow around a square cylinder of diameter $D$ with rounded corner modifications of radius $r$. The numerical method described in Section[2]
Figure 1: Schematic of the computational domain showing the (a) side view and (b) top view. The boundary conditions imposed on each boundary $\Gamma_i$ of the domain is specified.

Table 1: Results from the mesh convergence study carried out to ensure grid-independence of the solution for the case of flow past a square cylinder at $Re = 1 \times 10^5$. 

| Mesh No: | Mesh 1 | Mesh 2 | Mesh 3 | Mesh 4 |
|---------|--------|--------|--------|--------|
| Mesh Count | 633270 | 2574667 | 4946393 | 5291319 |
| $C_{j_{\text{m}}}$ | 2.76 | 2.25 | 2.33 | 2.33 |
| $C_{j_{\text{rms}}}$ | 0.21 | 0.27 | 0.23 | 0.20 |
| $C_{j_{\text{rms}}}$ | 1.40 | 1.15 | 1.30 | 1.33 |
was first validated against established numerical results for the case of flow past a sharp-corner square cylinder at \( \text{Re} = 1 \times 10^5 \). In this reference work \cite{35}, large-eddy simulations (LES) have been carried out to study the problem of the flow past a square cylinder over a wide range of Reynolds numbers. Figure 1a represents the front view of the 3D computational domain in the Cartesian coordinate system of size \( 30D \times 24D \times 4D \). The square cylinder is positioned at an upstream distance of \( 10D \) from the inlet and a downstream distance of \( 20D \) from the outlet boundary in the X direction and is at an equal distance of \( 12D \) from the top and bottom walls. The length of the cylinder is oriented along the Z direction as shown in Figure 1b.

To better characterise the flow, we define non-dimensional numbers like the Reynolds number \( \text{Re} = \rho U_0 D/\mu \), normalised corner radius \( r/D \), and blockage ratio \( B_f = D/L_y \), where \( D \) is the diameter of the square-cylinder, \( L_y = 24D \) is the width of the domain, \( \rho \) is the density of the fluid, and \( \mu \) is the dynamic viscosity. For the present case, the blockage ratio is 4.16%. A lower blockage ratio ensures that the solution is not affected by the domain boundaries. Figure 2 shows the geometrical comparison between the sharp and rounded corners for a square cylinder. We simulate 4 cases with different corner modifications. The normalised corner radii considered are \( r/D = 0.05, 0.10, 0.15, \) and \( 0.20 \) in addition to the sharp-corner case.

Figure 1 also presents the boundary conditions enforced on the domain boundaries. An uniform stream of fluid enters the domain at a velocity of \( U_0 \) along the X axis at the inlet boundary \( \Gamma^{\text{in}} \). \( \Gamma^{\text{out}} \) is a zero-gradient velocity and zero static pressure outlet boundary. At \( \Gamma^{\text{top}} \) and \( \Gamma^{\text{bot}} \), a symmetry boundary condition has been imposed. Since the cylinder is span-wise symmetric, \( \Gamma^{\text{right}} \) and \( \Gamma^{\text{left}} \) have a symmetry boundary condition too. In addition to the boundary conditions shown in Figure 1, a no-slip boundary condition has been enforced along the surface of the square cylinder. The values for \( k_0 \) and \( \omega_0 \) used in the inlet and outlet boundaries have been calculated based on the recommendation of \cite{36}.

A mesh convergence study has been carried out to ensure grid independence of the solution, and the results can be seen in Table 1. Based on these results, an unstructured polyhedral mesh of about 5.3 million elements has been selected. Inflation layers with a first-cell height of 0.0002 \( D \) have been provided along the surface of the cylinder to satisfy the requirement of \( y^+ \leq 1 \) for the \( k-\omega \) SST RANS model.

In the present work, all the simulations have been carried out on FLUENT, the commercial CFD solver by ANSYS. The solver employs a finite volume scheme to discretize
| Reference work [35] | Current Results | Error % |
|---------------------|-----------------|---------|
| $C_{d}^{\text{mean}}$ | 2.20            | 2.33    | 5.90   |
| $C_{d}^{\text{rms}}$  | 0.21            | 0.20    | 4.76   |
| $C_{l}^{\text{rms}}$  | 1.40            | 1.33    | 5.00   |

Table 2: Validation of the results obtained using the proposed numerical methodology against published LES results for flow past a square cylinder at $\text{Re} = 1 \times 10^{5}$ presented in [35].

Figure 2: Schematic of the side profiles for (a) the sharp corner case and (b) the rounded corner case where $r$ represents the radius of the corner modification on the square cylinder geometry.

The governing equations described in Section 2.1. The QUICK scheme [37] has been used for the spatial discretization, and a bounded second order time discretization with a timestep of $0.1D/\text{U}_0$ has been adopted for the time marching. A Strouhal number calculation indicates that the timestep used in the current work resolves each wavelength by about 70 points which is more than sufficient to ensure accuracy in time. The results obtained from these simulations are compared with the reference work [35] in Table 2. We observe that mean drag, fluctuating drag, and fluctuating lift coefficients match well with an average deviation of about 5%.

3. Results

A systematic series of DES simulations were performed for a Reynolds number of $\text{Re} = 1 \times 10^{5}$, considering four different normalised corner radii of $r/D = 0.05, 0.10, 0.15$, and 0.20 to understand the effect of corner modifications on the fluid dynamic parameters such as the drag coefficient, lift coefficient, Strouhal number, and the pressure coefficient. A non-dimensional time-step $\delta t \text{U}_0/D = 0.10$ was considered for the numerical simulations and the cases were averaged over 1500 time-steps.
3.1. Aerodynamic drag and pressure

Figures 3 and 4 summarise the effects of normalised corner radius on the drag and pressure coefficients for a square cylinder. In these figures, $r/D = 0$ corresponds to a sharp-corner square cylinder. We begin by analysing the drag force experienced by the cylinder. From Figure 3, we study the time-averaged drag coefficient ($C_{d,\text{mean}}$) for different cylinder configurations and compare them with the result obtained experimentally for a fully circular cylinder at the same Reynolds number as our simulations [38]. $C_{d,\text{mean}}$ decreases monotonically with an increase in $r/D$, with a steeper drop observed between $r/D = 0$ and $r/D = 0.10$ as compared to the latter half. We observe a decrease of almost 43% in $C_{d,\text{mean}}$ between the sharp-corner case and the rounded-corner case with $r/D = 0.20$. For the corner modified cylinders, the decrease in $C_{d,\text{mean}}$ is primarily due to a decrease in the size of the stagnation pressure region in the front of the cylinder and a smaller suction pressure in the wake which can be observed in the figures 4 and 5, detailed discussions on which have been presented later in this section. Comparing the above results with those obtained for a fully circular cylinder, we observe that the $C_{d,\text{mean}}$ for a sharp-corner square cylinder is 93.32% higher than that for the fully circular case. Similarly, the $C_{d,\text{mean}}$ for the corner modified cylinder with $r/D = 0.20$ is only 10.28% higher than its fully circular counterpart, thus indicating that the $r/D = 0.20$
configuration performs the best among the simulated cases. It is evident from the variation in the quantities with an increase in the $r/D$ value that a small change in the corner radius of the sharp corner square cylinder resulted in a significant drop in the drag experienced by it.

To explain the observations obtained in Figure 3, we investigate the distribution of pressure along the surface of the cylinder. Figures 4(a) and (b) represent the variation of the mean and fluctuating values of the pressure coefficient. In Figure 4(a), we can notice that for a sharp corner square cylinder, $C_p^{\text{mean}}$ has two valleys near the windward and leeward corners. The valleys represent local minima in the $C_p^{\text{mean}}$ values. For the sharp corner case, the valley is observed exactly at $45^\circ$ with respect to the X axis, measured from the centre of the cylinder. Comparing the pressure distribution for different $r/D$ configurations along the windward corner, we observe that the case with $r/D = 0.05$ has the largest dip in $C_p^{\text{mean}}$ value. The minima shifts upwards in value with an increase in the normalised corner radius. For the cylinder with $r/D = 0.05$, the minimum occurs at an angle less than $45^\circ$ and then shifts to the right along the $\theta$ axis. For square cylinders with $r/D = 0.15$ and 0.20, the valley is observed at an angle greater than $45^\circ$. In addition to this, we also observe that the width of the valleys at the frontal corner increase with an increase in the corner radii.

Along the leeward corner, a valley is only observed for the sharp cylinder and the $r/D = 0.05$ cases. The minimum is the largest for the sharp corner configuration, and is less pronounced for the $r/D = 0.05$ case. For the other corner modified cases, no valley is observed, thus indicating the absence of a suction region on the rear face of the cylinder. We note that the mean pressure at the leeward face ($\theta > 135^\circ$) rises with an increase in $r/D$. This leads to a lower pressure difference between the windward and leeward faces for higher values of $r/D$. Thus, the mean drag coefficient decreases with an increase in the normalised corner radius as observed in Figure 3.

We also study the effect of corner modifications on the unsteady aerodynamic loads experienced by the cylinder. Figure 4(b) illustrates the fluctuating values of the pressure coefficient ($C_p^{\text{rms}}$) to quantify the fluctuation amplitudes of the surface pressure distributions on the surface of the cylinder. In Figure 4(b), two peaks in the $C_p^{\text{rms}}$ values are observed along the surface of the cylinder that represent the pressure fluctuation amplitudes at the windward and leeward corners. For the square cylinder case with sharp corners, $C_p^{\text{rms}}$ has a larger peak at the leeward corner. It then gradually decreases along the leeward face of the cylinder. For the $r/D = 0.05$ configuration, the pressure fluctuation at the windward corner is higher than that at the leeward corner. We also observe a
Figure 4: Variation of mean and fluctuating pressure coefficient values along the surface of the cylinder for the sharp corner and rounded corner modified cases. Here $\theta$ represents the angle along the cylinder surface measured starting from the upstream stagnation point.
larger fluctuation amplitude along the rear face of the cylinder for the $r/D = 0.05$ case as compared to the others.

Apart from these, $C_{p}^{rms}$ along the side faces for the corner modified cases are much lower than that observed for the sharp corner case. Furthermore, for the corner modified configurations with $r/D = 0.10, 0.15, \& 0.20$, no peaks were observed at the leeward corner and they depart significantly from the trend observed for the sharp corner case. These observations indicate that drag reduction and suppression of unsteady aerodynamic forces are simultaneously realised by increasing the rounding radius of the edges of a square cylinder.

The significant effect of the normalised corner radius indicates a change in the flow patterns between $r/D = 0$ and $r/D = 0.20$. We study statistical flow fields in order to explain the characteristics of local pressure distribution observed in Figure 4. Figure 5 represents the span and time-averaged velocity streamline plots superimposed on the mean pressure coefficient contours for the sharp case and the four rounded corner modified cases. The flow topology for the five configurations differ significantly from each other. The most notable point is that the separation angle at the leading edge decreases with an increase in $r/D$. When the corner radius of the cylinder is increased, a large curvature is introduced in the streamlines near the windward corner. As a result of this curvature, the streamlines get closer to each other near the windward and leeward corners. This explains the valleys observed in Figure 4(a). The increase in curvature is also responsible for the increase in the width of the valleys for higher values of $r/D$. For the sharp corner square cylinder, the flow separates exactly at 45°. For the cylinder with $r/D = 0.05$, separation occurs early as the valley is observed at an angle less than 45°. The angle of separation increases for a subsequent increase in $r/D$, thus representing a corresponding delay in flow separation. We also observe a valley along the leeward corner, indicating flow separation caused by reversed flow from the recirculation region in the wake of the cylinder. The flow separation creates a small bubble which merges into the bigger recirculation bubble created due to the initial separation at the windward corner. With an increase in the $r/D$ value, smaller recirculation bubbles are created due to flow separation at the leeward corner as indicated by a decrease in the valley size and their subsequent absence in Figure 4(a). For the square cylinder case with $r/D = 0.20$, the flow creeps along the leeward corner and no bubble is formed. It can also be noted that with an increase in the normalised corner radius, the size of the recirculation region on the top and bottom walls of the cylinder decreases. The total size of the recirculation domain in the cross-flow direction also decreases. It is also accompanied by an increase
Figure 5: Velocity streamlines superimposed on mean pressure coefficient contours for (a) the sharp case and (b)-(e) the corner modified cases.
in the length of the recirculation region along streamwise direction in the wake of the cylinder. This narrowing of the recirculation region in the wake is responsible for the decrease in the mean pressure difference across the faces of the cylinder. Focusing on the distribution of time-averaged pressure around the cylinder, we observe that the magnitude of the stagnation pressure in front of the cylinder and the negative pressure behind the cylinder decrease with a corresponding increase in the $r/D$ value. We also notice a decrease in the width of the stagnation pressure envelope in front of the cylinder on increasing the rounding radius. A similar observation is made for the width of the negative pressure region in the wake of the cylinder.

To further explain our observations from Figure 5, we investigate the flow field in the wake of the cylinder. Figure 6 summarises the mean velocity profiles in the near wake region ($x/D = 0.75$ and $1.00$) and the far-wake region ($x/D = 1.50$ and $2.50$). Here, $x/D$ is the non-dimensional distance in the streamwise direction measured from the centre of the cylinder. The outward bulge in the profiles denotes region of velocity deficit where fluid slows down due to the presence of a recirculation region. In the near-wake region, we find noticeable cusps at $y/D = 0$ and the size of these cusps decrease as we go further downstream before completely disappearing for $x/D = 1.50$ and $2.50$. We observe that these cusps disappear quickly for the sharp corner case than in comparison to the corner modified cases, thus indicating a faster recovery of velocity deficit by the sharp corner cylinder in the near-wake region. However, in comparison to the near-wake region, we note that size of the outward bugle decreases at a faster rate for the corner modified cases as compared to their sharp corner counterpart for $x/D = 1.50$ and $2.50$. Therefore, they are able better recover the velocity deficit in the far-wake region, despite having a longer recirculation region. This is primarily due to higher value of fluid momentum observed in the recirculation region of the corner modified cases due to the narrowing of the recirculation region in the Y direction.

3.2. Aerodynamic lift and Strouhal number

In this section, we examine the variation of the lift coefficient and the Strouhal number to study the effect of corner modifications on the characteristics of the wake. We start by considering the root mean squared (RMS) values of instantaneous lift coefficient ($C_{l}^{rms}$) presented in Figure 7. $C_{l}^{rms}$ represents the effective force of the fluid on the surface of the cylinder that causes oscillations in the cross-flow direction. $C_{l}^{rms}$ is highest for the sharp corner case and decreases monotonically as the $r/D$ value is increased for the corner modified cases. The high value of $C_{l}^{rms}$ for the sharp corner square cylinder can be attributed to the height of the wake in the cross-flow direction. There is a nearly
Figure 6: Profiles of mean velocity magnitude plotted along the vertical direction at different points in the wake of the square cylinders
55% decrease in lift coefficient between the sharp corner and the $r/D = 0.20$ case. In particular, a sharp drop is observed between the $r/D = 0.05$ and $r/D = 0.10$ cases. Comparing the square cylinder cases to the fully circular cylinder [38], we note that $C_{l_{rms}}$ for the circular cylinder is 77.4% lower than that for the sharp corner case, and approximately 49.82% lower than that of the corner modified case with $r/D = 0.20$.

A spectral analysis using the Fast Fourier Transform (FFT) was performed on the time history of the lift coefficient to obtain the Strouhal number. It is the variation of the non-dimensionless frequency, and is denoted by $St = fD/U_0$. Here, $f$ is the vortex shedding frequency, $D$ is the characteristic dimension of the flow and $U_0$ is the freestream velocity. The effect of the normalised corner radius on the Strouhal number is presented in Figure 8. The spectral analysis results for the square cylinder cases are compared with those obtained for a circular cylinder at Re = $1 \times 10^5$ in [38]. In Figure 8, we observe that $St$ increases monotonically for the square cylinder cases, with the sharp-corner square cylinder having the lowest value. This is primarily due to flow separation at the frontal edge of the cylinder that generates a wider recirculation region in the wake. A wider wake leads to a lower vortex shedding frequency. Compared to the sharp corner case, the $St$ value for $r/D = 0.20$ configuration increased by 22.4%.
This increase can be attributed to the delay in flow separation at the rounded corners, which give rise to narrower, but longer recirculation regions in the wake of the cylinders as observed in Figure 5. Therefore, we can infer that the vortex shedding frequency is directly proportional to the size of the wake in the cross flow direction. However, it is noted that these values are still low in comparison to the value obtained for a fully circular cylinder [38]. The Strouhal number for the $r/D = 0.20$ case is 25.8% lesser than the results obtained for its fully circular counterpart. Figure 9 displays the energy spectra of the fluctuating lift coefficient for the various square cylinder cases. The peaks represent the energy of the shed vortices in the wake. We note a decrease in the magnitude of the peaks with an increase in the $r/D$ value, but this decrease is not monotonous. Instead, we observe an increase in the strength of vortex shedding when the normalised corner radius is increased from 0.10 to 0.15. The $r/D = 0.20$ case has the weakest vortices with an approximately 96% decrease in the strength in comparison to the sharp corner case. Furthermore, the peaks shift right along the X axis with an increase in the $r/D$ value. The X axis represents the dominant frequency of vortex shedding in the wake of the cylinder. We notice that with an increase in the rounded corner radius, the frequency of vortex shedding increases. This explains the increase in the Strouhal number for the corner modified cases observed in Figure 5.
The turbulent statistics of the wake are further studied using the span-averaged fluctuating stream-wise velocity contours presented in Figure 10. We observe that for all cases, the velocity fluctuation next to the side walls is greater than that in the wake region. However, the magnitude of peak velocity fluctuations decrease with an increase in the $r/D$ value. Furthermore, as we move away from the body, the effect of streamwise velocity diminishes. It can be observed in the form of a decreasing region of influence for an increase in the rounding radius. The peaks are a result of the large deflection angles of the separated shear layers at the frontal edge of the cylinder. Their magnitude decreases as the wake becomes narrower in the cross-flow direction. A reduced fluctuating stream-wise velocity and a reduced region of influence leads to a decrease in the root mean square (RMS) values for the force coefficients, as has been observed in the case of $C_l^{\text{rms}}$ in Figure 7.

4. Conclusion

This work presents a comprehensive analysis of the effects of rounded corner modifications $0 \leq r/D \leq 0.20$ on the flow past a square cylinder at a high Reynolds number of $Re = 1 \times 10^5$. 3D simulations have been carried out using the IDDES
Figure 10: Span-averaged fluctuating stream-wise velocity contours for (a) the sharp case and (b)-(e) the corner modified cases.
formulation and validation of DES results against established LES results for the flow around a sharp-corner square cylinder is provided. The conclusions can be summarised as follows:

1. Increasing corner radius leads to a steady decrease in separation angle. It is accompanied reduction in the width of the recirculation region near the leading edge and an increase in the length of the wake in the streamwise direction.

2. The pressure difference between the windward and leeward sides of the cylinder reduces as a consequence of corner modifications thus leading to a reduction in mean pressure drag.

3. There is a substantial decrease in the mean and fluctuating values of force coefficients observed between the sharp corner case and the ones with rounded corner modifications; more specifically a 43.2% decrease in $C_{d\text{mean}}$, a 62.1% decrease in $C_{d\text{rms}}$ and a 55.5% decrease in $C_{l\text{rms}}$ between the sharp case and the $r/D = 0.20$ case.

4. The narrowing of the recirculation region along the streamwise direction results in an increase in the vortex shedding frequency in the wake of the corner modified cases.

5. A longer recirculation region for the corner modified cases leads to a faster recovery of velocity deficit in the far wake region.

The technical implications of the observations above are important in studies that aim to decrease wind loading on bluff bodies like short buildings, which are subject to high Reynolds number flows. This numerical simulation provides evidence of significantly reduced aerodynamic force coefficients upon increasing the corner modification radius.

5. Acknowledgement

The authors would like to acknowledge the financial support received from LIGO India to carry out the following study.

References

[1] H Kawai. Effect of corner modifications on aeroelastic instabilities of tall buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, 74-76:719–729, 1998.

[2] N. Shiraishi, M. Matsumoto, H. Shirato, and H. Ishizaki. On aerodynamic stability effects for bluff rectangular cylinders by their corner-cut. *Journal of Wind Engineering and Industrial Aerodynamics*, 28:371–380, 1988.
[3] David Rehacek, Tomáš Khel, Josef Kucera, Jan Vopavil, and Martin Petera. Effect of windbreaks on wind speed reduction and soil protection against wind erosion. *Soil and Water Research*, 12:128–135, 2017.

[4] W. M. Cornelis and D. Gabriels. Optimal windbreak design for wind-erosion control. *Journal of Arid Environments*, 61:315–332, 2005.

[5] B. Tsai and B. Shiau. Experimental study on the flow characteristics for wind over a two-dimensional upwind slope escarpment. *Journal of Marine Science and Technology*, 2011.

[6] K. T. Tse, P. A. Hitchcock, K. C. S. Kwok, S. Thepmongkorna, and C. M. Chanb. Economic perspectives of aerodynamic treatments of square tall buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, 97:455–467, 2009.

[7] K. C. S. Kwok, P. A. Wilhelm, and B. G. Wilkie. Effect of edge configuration on wind-induced response of tall buildings. *Engineering Structures*, 10:135–140, 1988.

[8] Young-Moon Kim, Ki-Pyo You, and Nag-Ho Ko. Across-wind responses of an aeroelastic tapered tall building. *Journal of Wind Engineering and Industrial Aerodynamics*, 96(8-9):1307–1319, 2008.

[9] Md. Naimul Haque. Effectiveness of corner modification to optimize aerodynamic responses of square cylinder. *Journal of Physics: Conference Series*, 1519:12–18, 2020.

[10] M. S. Aswathy, K. K. Amrita, C. M. Hariprasad, and R. Ajith Kumar. Near-Wake Flow Structures of a Corner-Chamfered Square Cylinder at Higher Harmonic Excitations. In *Volume IA: Symposia, Part 2*, page V01AT12A004. American Society of Mechanical Engineers, 2015.

[11] Yi Li, Chao Li, Qiu-Sheng Li, Qian Song, Xuan Huang, and Yong-Gui Li. Aerodynamic performance of CAARC standard tall building model by various corner chamfers. *Journal of Wind Engineering and Industrial Aerodynamics*, 202:104197, 2020.

[12] Noel K. Delany and Norman E. Sorensen. Low-speed drag of cylinders of various shapes. 1953.

[13] B. E. Lee. The effect of turbulence on the surface pressure field of a square prism. *Journal of Fluid Mechanics*, 69:263–282, 1975.
[14] S Okamoto and N Uemura. Effect of rounding side-corners on aerodynamic forces and turbulent wake of a cube placed on a ground plane. *Experiments in Fluids*, 11(1):58–64, 1991.

[15] T. Tamura and T. Miyagi. The effect of turbulence on aerodynamic forces on a square cylinder with various corner shapes. *Journal of Wind Engineering and Industrial Aerodynamics*, 83:135–145, 1999.

[16] J. C. Hu, Y. Zhou, and C. Dalton. Effects of the corner radius on the near wake of a square prism. *Experiments in Fluids*, 40:106–118, 2006.

[17] Rajeev K. Jaiman, Subhankar Sen, and Pardha S. Gurugubelli. A fully implicit combined field scheme for freely vibrating square cylinders with sharp and rounded corners. *Computers & Fluids*, 112, 2015.

[18] Luigi Carassale, Andrea Freda, and Michela Marrè-Brunenghi. Experimental investigation on the aerodynamic behavior of square cylinders with rounded corners. *Journal of Fluids and Structures*, 44:195–204, 2014.

[19] Liuliu Shi, Geer Yang, and Shichuan Yao. Large eddy simulation of flow past a square cylinder with rounded leading corners: A comparison of 2d and 3d approaches. *Journal of Mechanical Science and Technology*, 32:2671–2680, 2018.

[20] Wei Zhang and Ravi Samtaney. Low-Re flow past an isolated cylinder with rounded corners. *Computers & Fluids*, 136, 2016.

[21] Sajjad Miran and Chang Hyun Sohn. Influence of incidence angle on the aerodynamic characteristics of square cylinders with rounded corners: A numerical investigation. *International Journal of Numerical Methods for Heat and Fluid Flow*, 26:269–283, 2016.

[22] Yong Cao and Tetsuro Tamura. Supercritical flows past a square cylinder with rounded corners. *Physics of Fluids*, 29(8):85–110, 2017.

[23] Yong Cao and Tetsuro Tamura. Shear effects on flows past a square cylinder with rounded corners at $\text{Re} = 2.2 \times 10^4$. *Journal of Wind Engineering and Industrial Aerodynamics*, 174:119–132, 2018.

[24] Yong Cao and Tetsuro Tamura. Aerodynamic characteristics of a rounded-corner square cylinder in shear flow at subcritical and supercritical reynolds numbers. *Journal of Fluids and Structures*, 82:473–491, 2018.
[25] S. S. Dai, B. A. Younis, and H. Y. Zhang. Prediction of turbulent flow around a square cylinder with rounded corners. *Journal of Offshore Mechanics and Arctic Engineering*, 139:031804, 2017.

[26] Doohyun Park and Kyung-Soo Yang. Flow instabilities in the wake of a rounded square cylinder. *Journal of Fluid Mechanics*, 793:915–932, 2016.

[27] Yoshiyuki Ono and Tetsuro Tamura. LES OF FLOW AROUND A CIRCULAR CYLINDER IN THE CRITICAL REYNOLDS NUMBER REGION. 2008.

[28] I Rodriguez, O Lehmkuhl, J Chiva, R Borrell, and A Oliva. On the flow past a circular cylinder from critical to super-critical Reynolds numbers: Wake topology and vortex shedding. *International Journal of Heat and Fluid Flow*, 55:91–103, 2015.

[29] Seong Mo Yeon, Jianming Yang, and Frederick Stern. Large-eddy simulation of the flow past a circular cylinder at sub- to super-critical Reynolds numbers. *Applied Ocean Research*, 59:663–675, 2016.

[30] Junting Chen. *Effect of Aspect Ratio on the Flow Structures Behind a Square Cylinder*. PhD thesis, University of Windsor, 2018.

[31] P. R. Spalart, S. Deck, M. L. Shur, K. D. Squires, M. Kh. Strelets, and A. Travin. A new version of detached-eddy simulation, resistant to ambiguous grid densities. *Theoretical and Computational Fluid Dynamics*, 20:181, 2006.

[32] F. R. Menter and M. Kuntz. Adaptation of eddy-viscosity turbulence models to unsteady separated flow behind vehicles. pages 339–352, 2004.

[33] F. R. Menter. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32(8):1598–1605, 1994.

[34] Mikhail L. Shur, Philippe R. Spalart, Mikhail Kh. Strelets, and Andrey K. Travin. A hybrid rans-les approach with delayed-des and wall-modelled les capabilities. *International Journal of Heat and Fluid Flow*, 29:1638–1649, 2008.

[35] A. Sohankar. Flow over a bluff body from moderate to high reynolds numbers using large eddy simulation. *Computers & Fluids*, 35:1154–1168, 2006.

[36] Philippe Spalart and Christopher Rumsey. Effective inflow conditions for turbulence models in aerodynamic calculations. *Aiaa Journal - AIAA J*, 45:2544–2553, 2007.
[37] T Hayase, J.A.C Humphrey, and R Greif. A consistently formulated quick scheme for fast and stable convergence using finite-volume iterative calculation procedures. *Journal of Computational Physics*, 98:108–118, 1992.

[38] Günter Schewe. On the force fluctuations acting on a circular cylinder in crossflow from subcritical up to transcritical Reynolds numbers. *Journal of Fluid Mechanics*, 133:265–285, 1983.