HEAT TRANSFER AND FLUID FLOW PROPERTIES ACROSS A BLUFF BODY AT MODERATE REYNOLD NUMBER

Shehnaz Akhtar [1] & Farzeen Shahid[2]
[1, 2] Department of Mechanical Engineering, COMSATS Institute of Information Technology, Sahiwal
Email: Shehnaz@ciitsahiwal.edu.pk[1], farzeen@ciitsahiwal.edu.pk[2]

ABTSRACT

Fluid flow past bluff bodies (such as circle or square cylinder) is of great concern in previously published literature because of its in engineering applications. TPL (tension leg platform), wires, towers, suspension bridges off-shore structures, tall buildings, piers all of these are constantly subjected to variable fluid flow load. When fluid flow across these objects/structures flow is separated from the main flow direction and creates a large low pressure zone (wake) downstream of these objects. Vortices are shed in the wake region thus causing velocity fluctuations in the low pressure zone and give rise to unsteady drag and lift forces. Now if the vortex shedding frequency matches the natural frequency of the object the phenomenon of resonance will occur that will cause vibration in the structure or ultimately it will come to failure. Heat transfer across bluff bodies which are subjected to vortex shedding has many applications including, electronic cooling components and heat exchanger.

Keywords: *Reynold number, Vortex shedding, Drag and lift coefficient, recirculation, Karman Vortex Street, cylinder, Nusselt number*

1. Introduction

A bluff body is that body in which stream under ordinary conditions isolates from a substantial segment of body surface. Thus making noteworthy downstream wake region. Over the previous years the stream around bluff bodies represented a fluid mechanics issues which generally include three shear layers named as Boundary layer, an isolating free shear layer and a wake each with various procedures of causing instabilities as the Reynold number (Re) is increased. When Reynolds number is very low ($Re \leq 1$), flow traces out the cylinder contours. When the Reynolds number is 1.15, flow separation happens from the base point of the structure and when the Reynolds number is between 5 and 100 ($5 \leq Re \leq 100$) flow separates from the rear sharp corners and accompanies a pair of steady symmetric vortices in the near wake. Stream-wise length of the vortices increases linearly with Reynolds number, approaching at critical onset Reynolds number (Re_{crit} ~50), where the twin vortex arrangement becomes unstable resulting a time periodic oscillating wake and a staggered vortex street is formed known as von Karman Vortex Street. At higher Reynolds number ($Re \geq 100$) flow separates from the upstream corners and a 3D transition is developed at around Reynolds number 200 (Re_{cr2}), and three dimensional flow and turbulence effects appear. Within the laminar regime ($Re_{cr1} \leq Re \leq Re_{cr2}$), the vortex shedding is characterized by very well defined frequency which increases with Reynolds number. The behavior of the fluid flow defines the performance of the device, forces**,** vibration or heat transfer rates for various engineering application. Bluff body induced instability in the flow and mixing of the flow can be used to improve heat transfer and mass transfer between the body and surrounding.

2. Bluff Body Aerodynamics

Figure 1 shows flow past a square bluff body for Reynold number higher than 150.In this figure U describe the free stream velocity, P_0 represent free stream mean dynamic pressure , F_L and F_D represent lift and drag force respectively which are exerted by fluid on the body.

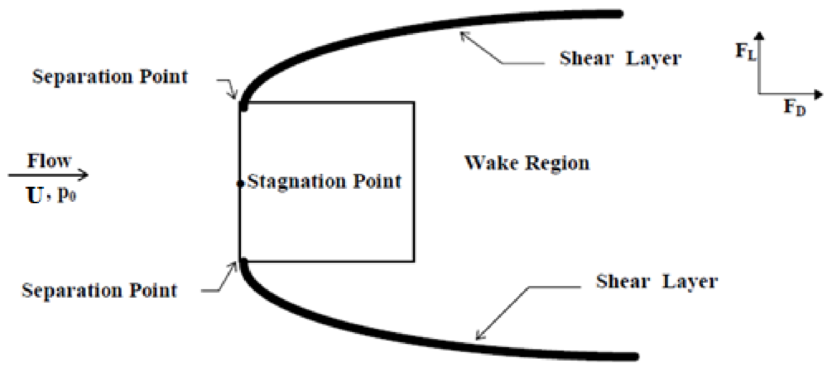

Figure 1

The pressure on the surface of the body *(P*) is usually referenced to P_0. Therefore, from Bernoulli's equation, if the velocity at any point over the surface of the body is greater than the free stream velocity, the pressure will be less than P_0 (negative). Conversely, the pressure will be positive in areas with a velocity less than the free stream velocity.

The separation point refers to the position on a body immersed in a flow where the boundary layer separates from the body. The location of separation on a bluff body with a surface discontinuity, as shown in Figure 1 is less complicated than that for a bluff body with a continuous surface, such as circular cylinder. The boundary layer grows from the stagnation point and separates at a surface discontinuity (sharp corners) to form what is known as a separated boundary layer or a separated shear layer. For Re<150, fluid may follow the contours of the body and consequently separation may not occur.

3. Numerical Analysis

In past, vast majority of studies on the problem of wake development and vortex shedding behind a bluff body in free-stream flows at low to moderate Re have been investigated both numerically and experimentally by several authors.

A.K. Saha *et al* (2003) worked on the three dimensional flow on the square cylinder. The Reynold number in his study ranges from 150 to 500. He concluded that the 3D transition occurs at the Reynolds number ranges from 150 to 175. At Reynolds number of 175, dislocations of vortex were observed in the flow and at Reynolds number of 250, the flow is much finer as compared to the previous one. [1]. Zhang Ling et al (2008) studied flow simulation around a square cylinder using turbulent model. He concluded that if the modification is made in the generation of turbulence kinetic energy, then the simulation is more successful then modifying the turbulence near the wall. [2] O. Almeida *et al* (2008) performed 2D calculations of the flow past a square cylinder by employing LES model and investigated the effects of the rectangular cylinder aspect ratio on the wake dynamics. LES model presented a good agreement of numerical results with the experimental data. [3]

T Tamura and T Miyagi investigated numerically 3D flow past a square cylinder with different corner shapes at Re $=10^4$ by employing finite difference scheme. The results indicate that a slight change in corner shape results reduction in aerodynamic coefficient. The reduction in drag coefficient was found almost 60% of the original value [4]. Sohankar *et al* performed numerically worked on 2D and 3D flow past square cylinder and concluded that transition from 2D to 3D occurs between the ranges of Reynolds number 150-200. The main difference between the fluid flow across square and circular cylinder is that the separation takes place from the upstream corners for square cylinder [5]. Stefan Schmidt & Frank Thiele applied many numerical techniques, namely DES and RANS for fluid flow over wall mounted cubes and for fluid flow across square cylinder and showed that the more the unsteady motion, DES will be more effective than RANS [6].

C.Dalton and W. Zheng contemplated numerically 2D viscous laminar liquid stream over square and diamond cylinder, with sharp and adjusted corners at Reynold number 250 and by using the finite difference technique. The elliptic partial differential condition for stream-function and vorticity in the transformed plane is explained by the multigrid iteration method. Vortex shedding phenomenon variation in drag coefficient C_D and Strouhal number (S_t) were observed. An obvious reduction in the aerodynamic forces was found in both square and diamond cylinder. [7].

Atul Sharma and Eswara carried out 2D numerical investigation of forced convective heat transfer of cross flow for stationary square cylinder in the laminar range of Reynold number (*Re)*. They noted that onset of vortex shedding occur between Re 50 to 55.Further more by increasing Reynold number increase in average heat transfer was found [8]. Islam.S. U *et al.* performed 2D and 3D numerical simulations of cross fluid flow past four square cylinders which were in-line arranged at a *Re*number of 100000 and 200 respectively. By the two dimensional studies three distinct flow patterns; balanced shielding flow, squirm shielding flow and vortex shedding were found and it was observed that transformation from one flow pattern to other increases the amplitude of the maximum fluctuating pressure on the downstream cylinder [9]. O.Almeida *et al* performed 2D numerical simulation of fluid flow across square cylinder by employing LES model and investigated the effects of the rectangular cylinder aspect ratio on the wake dynamics. LES model presented a good agreement with previously published experimental and numerical results [10].

Andrea Mola *et al.* worked on the three dimensional simulation of fluid flow across square cylinder with rounded and sharp corners. They studied the effect of aspect ratio by using Reynold number of 10^5. A conclusion is made that over the finite length cylinder, low frequency components are elementary aspect of the flow and are associated with the free end of cylinder. This causes a loss of synchronization results in variations of the mean drag coefficient along the cylinder. These variations are more significant in the case of cylinders with sharp corners than in the case of those with rounded corners [11]. Gera.B *et al.* carried out computational fluid dynamics analysis for flow around square cylinder for Reynolds number ranging from 50 to 250.Results showed that the vortex shedding starts between *Re* number 50 and 55 and it is exhibited by a using dominated frequency for Re>55.Their calculated results of aerodynamic coefficients showed good agreement with other reported results [12]. Dipankar Chatterjee *et al.* performed systematic study for the flow around a row of five square cylinders placed in a side-by-side arrangement and normal to the oncoming flow at a Reynolds number of 150. Depending on the separation ratio the following flow patterns were observed: a flip-flopping pattern, in-phase and anti-phase synchronized pattern and non-synchronized pattern. [13]

M. Sajjad and C.H. Sohn examined numerically stream past a square chamber with various corner radius and angle of incidence at Reynolds number 500 and presumed that corner radius and angle of incidence is found to essentially impact the stream qualities around the cylinder which incorporates drag and lift coefficients, instantaneous velocity and vorticity and Strouhal number. The flow around square cylinder with cut-corners at the front edges is explored utilizing particle image velocimetry. It is discovered that reduction in drag can be accomplished for the cut-corner dimensions. The mechanism for the reduction in drag is investigated on the statistical and structural parts of the stream. In the wake of compromising, the variance force of the wake is debilitated, the length of the distribution area behind the square cylinder is expanded, while the width of the wake diminishes. It is discovered that the drag coefficient is directly proportional to the minimum width of wake, and the Strouhal number St is inversely proportional to the minimum width of wake. It is observed that the lessened wake width is because of the blocked separation over the side surfaces for the cylinder with cut-corners at the front edges. [14]

4. Experimental Analysis

S. Manzoor *et al*. did experimental investigated on vortex induced vibrations of a square cylinder in a wind tunnel and measured the growth rates. Growth-rate results showed a sharp increase at the beginning of the lock-in, with a maximum for a reduced velocity. A slight hysteretic effect had also been highlighted in the mode switch area, i.e. for a reduced velocity closed to 9.5. [15]. P.W. Bearman *et al* studied experimentally the impact of corner radii on hydrodynamic forces experienced by square and diamond cylinder in oscillatory flow over the Keulegan Carpenter number ranges 1 to 100 and *Re* number in the range of 200 to 2 x 10^4. Coefficient of drag in oscillatory fluid flow was found very dependent on corner radius as compared to in steady flow. For constant Keulegan Carpenter number coefficient of drag of a diamond cylinder decreases linearly with the increase in corner radius and reduction in coefficient of drag was visible at low Keulegan Carpenter numbers [16].

K. C. S. Kwok *et al. C*onducted wind tunnel model tests to study the effect of edge shapes on the flow induced response of rectangular tall structures at wind velocity 4 to 20. Results showed that modified corners results visible reduction in wind response [17]. T. Tamura & T. Miyagi performed experimental investigation on the effects of corner shape, turbulence intensity in oncoming flows, and the effect of 3D on the averaged and fluctuating values of lift and drag forces. Their results showed that separated shear layers come closer to side surface with corner-cutting and corner rounding, thus enhancing reattachment and reduction in drag force [18].

Rahnama *et al.* studied convective heat transfer coefficient (*Nu*) from a rectangular cylinder at aspect ratio 0.5,1 (square cylinder) and 2 built in a channel at Reynolds number $50 \leq Re \leq 200$ by employing finite volume method. Results showed that increasing aspect ratio decreases mean total Nusselt number, whereas increasing *Re* number from 100 to 200 increases Nusselt number [19].

J. C. Hu *et al.* investigated experimentally the wake of square cylinder with different corner radii based on particle imaging velocimetry (PIV), laser Doppler anemometry (LDA) and hot wire measurements at Reynolds number of

2600 and 6000. Four square cylinder, i.e., r/d=0 (square cylinder), 0.157, 0.236, 0.5 (circular cylinder), where r is the radius of corner and d is the characteristic dimension of the body were investigated [20]. Guo Shen He *et al.* studied the effect of front corner shapes of a square cylinder on aerodynamic forces by using particle image velocimetery.They noted that corner cut at front causes the flow to go around the surface of the cylinder and increase recirculation of the flow behind the cylinder. They noticed reduction in drag coefficient after cutting the front corners of the cylinder [21].

Figure 2 shows the different drag reduction method for square cylinders [21]

a. Square cylinder with sharp corners
b. Variation of breath to width ratio b/D
c. Square cylinder with round or chamfered edge
d. Square cylinder with cut corner at the front edge

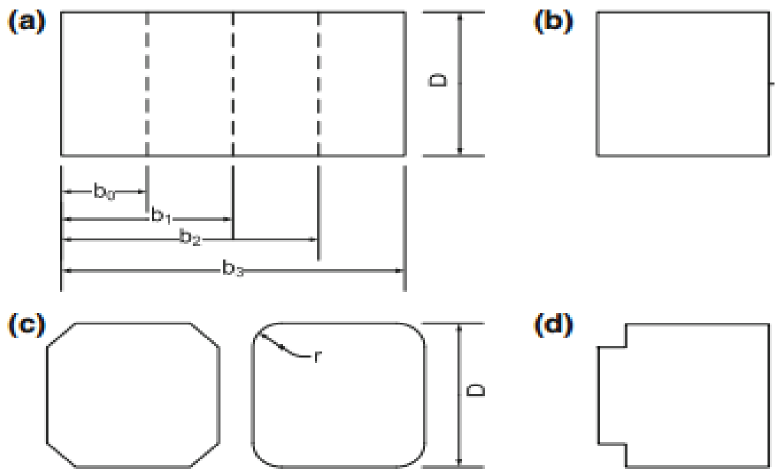

Figure 2: Drag reduction method for square cylinders [21]

5. Conclusion

Most of the reviewed studies (numerical and experimental) showed that at low Reynold number for the case of bluff bodies the flow separates from the upstream corners leaves a large separated region behind the cylinder and shows higher value of drag coefficient. However For modified-corners case (i-e Round chamfered), flow separation takes place from upstream corners, but the fluid goes around the side surface and tries to follow the contours of the body. It has been observed that for modified corners the recirculation of flow facilitates the fluid to go around the surface of the cylinder. These will results reduction in the wake width. For rounded corners the wake width becomes narrow with the increase in Reynold number that will cause reduction in drag. By increasing Reynold number, Nusselt number increases for sharp corners (i-e square cylinder) However, Nusselt number show higher values for modified corners with the increase in Reynold number. This is well explained after modifying the corners the fluid go around the side surfaces of the cylinder. The fluid surface contact is increased after modifying the corners that will increase the heat transfer.

6. References

[1] Saha, A. K., G. Biswas, and K. Muralidhar. "Three-dimensional study of flow past a square cylinder at low Reynolds numbers." *International Journal of Heat and Fluid Flow* 24.1 (2003): 54-66

[2] Hu, J. C., Y. Zhou, and C. Dalton. "Effects of the corner radius on the near wake of a square prism." *Experiments in Fluids* 40.1 (2006): 106-118.

[3] Almeida, O., S. Mansur, and A. Silveira-Neto. "On the flow past rectangular cylinders: physical aspects and numerical simulation." *Therm. Eng* 7.1 (2008): 55-64.

[4] Tamura, T. E. T. S. U. R. O., T. Miyagi, and T. Kitagishi. "Numerical prediction of unsteady pressures on a square cylinder with various corner shapes." *Journal of Wind Engineering and Industrial Aerodynamics* 74 (1998): 531-542.

[5] Sohankar, Ahmad, C. Norberg, and L. Davidson. "Simulation of three-dimensional flow around a square cylinder at moderate Reynolds numbers."*Physics of Fluids (1994-present)* 11.2 (1999): 288-306.

[6] Franke, R., W. Rodi, and B. Schönung. "Numerical calculation of laminar vortex-shedding flow past cylinders." *Journal of Wind Engineering and Industrial Aerodynamics* 35 (1990): 237-257.

[7] Dalton, C., and W. Zheng. "Numerical solutions of a viscous uniform approach flow past square and diamond cylinders." *Journal of fluids and structures* 18.3 (2003): 455-465.

[8] Sharma, Atul, and V. Eswaran. "Heat and fluid flow across a square cylinder in the two-dimensional laminar flow regime." *Numerical Heat Transfer, Part A: Applications* 45.3 (2004): 247-269.

[9] Islam, S. Ul, et al. "Numerical simulation of flow past rectangular cylinders with different aspect ratios using the incompressible lattice Boltzmann method." *Journal of mechanical science and technology* 26.4 (2012): 1027-1041.

[10] Almeida, O., S. Mansur, and A. Silveira-Neto. "On the flow past rectangular cylinders: physical aspects and numerical simulation." *Therm. Eng* 7.1 (2008): 55-64.

[11] Mola, Andrea, Giancarlo Bordonaro, and Muhammad R. Hajj. "Low-frequency variations of force coefficients on square cylinders with sharp and rounded corners." *Journal of structural engineering* 135.7 (2009): 828-835.

[12] Gera, B., Pavan K. Sharma, and R. K. Singh. "CFD analysis of 2D unsteady flow around a square cylinder." *International Journal of applied engineering research* 1.3 (2010): 602.

[13] Dipankar Chatterjee, Gautam Biswas, Sakir Amiroudine, 2010. Numerical simulation of flow past row of square cylinders for various separation ratios.Computers & Fluids 39:49-59

[14] Sajjad, M., and C. H. Sohn. "Numerical study of flow past a square cylinder with corner curvature at incidence." *Applied Sciences and Technology (IBCAST), 2014 11th International Bhurban Conference on*. IEEE, 2014.

[15] Manzoor, S., Pascal Hémon, and Xavier Amandolèse. "Vortex induced vibrations of a square cylinder in a wind tunnel." *ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting collocated with 8th International Conference on Nanochannels, Microchannels, and Minichannels*. American Society of Mechanical Engineers, 2010.

[16] Bearman, P. W., et al. "The influence of corner radius on the forces experienced by cylindrical bluff bodies in oscillatory flow." *Applied Ocean Research* 6.2 (1984): 83-89.

[17] Kwok, K. C. S. "Effect of building shape on wind-induced response of tall building." *Journal of Wind Engineering and Industrial Aerodynamics* 28.1-3 (1988): 381-390.

[18] Tamura, Tetsuro, and Tetsuya Miyagi. "The effect of turbulence on aerodynamic forces on a square cylinder with various corner shapes."*Journal of Wind Engineering and Industrial Aerodynamics* 83.1 (1999): 135-145.

[19] Rahnama, Mohammad, Seyed-Majid Hashemian, and Mousa Farhadi. "Forced convection heat transfer from a rectangular cylinder: effect of aspect ratio." *Proceedings*. 2005.

[20] Hu, J. C., Y. Zhou, and C. Dalton. "Effects of the corner radius on the near wake of a square prism." *Experiments in Fluids* 40.1 (2006): 106-118.

[21] He, Guo Sheng, Nian Li, and Jin Jun Wang. "Drag reduction of square cylinders with cut-corners at the front edges." *Experiments in fluids* 55.6 (2014): 1-11.