International Journal of Emerging Engineering Research and Technology
Volume 6, Issue 10, 2018, PP 24-33
ISSN 2349-4395 (Print) & ISSN 2349-4409 (Online)

Numerical Analysis of Wake Mode and Force Statistics for Flow over Two Rectangular Rods at Different Reynolds Numbers

R. Manzoor[1,2], S.U. Islam[1], S. Batool[2], A. Anwar[2], P. Akhtar[2]

[1]*Mathematics Department, COMSATS Institute of Information Technology, Islamabad 44000, Pakistan*

[2]*Mathematics Department, SBK Women's University, Quetta, Pakistan*

***Corresponding Author:** R. Manzoor, Mathematics Department, SBK Women's University, Quetta, Pakistan*

ABSTRACT

This paper presents a numerical simulation of flow over two rectangular rods with an inline arrangements in a free stream. The basic purpose of this study is to systematically investigate the influence of Reynolds number and gap spacing on flow behavior. The Reynolds number is chosen within the range 80-200 with gap spacing having the values g = 0.5-6. The obtained results are compared with those available in literature for purpose of code validation and found to be a good agreement. Four different types of flow modes were observed, that are (i) Steady flow mode (SF), (ii) Quasi steady flow mode (QS), (iii) Fully generated single slender body flow mode (FGSSB) and (iv) Fully generated irregular vortex shedding flow mode (FGIVS). The steady flow mode is obtained for $0 \leq g \leq 3$ at $80 \leq Re \leq 200$ and $4 \leq g \leq 6$ at $80 \leq Re \leq 100$. The physical parameters such as mean drag force, Strouhal number and root mean square values of drag and lift coefficients are also computed as a function of Re at various gap spacing. It is observed that the physical parameters strongly depend on flow modes and their behavior change due to change in flow phenomena. Furthermore, it is examined that the maximum mean drag force occurs at (Re, g) = (80, 6) due to negligible influence of both rods to one another because of sufficient gap spacing and its value continually decreases due to increment in Reynolds number. The effect of thrust is also observed for downstream rod at Re ≥ 150 for g = 0.5 – 1.5.

Keywords: *Control of vortex shedding; Mean drag force; Lattice Boltzmann method; Inline rectangular rods.*

INTRODUCTION

The researches based on flow over a single or multiple objects that may be square, circular or rectangular have been done since the forepart of last century and the flow passed through these objects is greatly influenced by Reynolds number (Re = $U_\infty d/\nu$) , aspect ratio (AR = l/w) and gap spacing (g = s/d, where d is the size of rod, s is the space between two rods, U_∞ is the inflow velocity, l is length, w is width of rod and v is kinematic viscosity) between more than one rod. There are many experimental and numerical studies are available in literature that based on the influence of Reynolds numbers and gap spacing for flow around an object Duta et al., 2004; Gera, 2010 and Sohankar et al., 1995.An experimental study of flow over a rectangular rod was done by Okajima (2006) in a wind tunnel. The variation in Strouhal number with the ratio of width-to-height was observed at the range of Re between 70 and 20000. There may also seem a noticeable change in flow mode due to discontinuity in values of Strouhal number. The Bearman and Trueman (1972) performed an experiment for flow over a rectangular rod at various Reynolds numbers and aspect ratios.

They examined that mean drag coefficient increases at Re $= 0.13 \times 10^3$ and AR = 0.62, whereas, St shows a discontinuous jump at AR = 2.8. Shadaram *et al*., (1980) conducted an experimental study on flow over a rectangular rod placed in a channel by considering Re between 8600 and 173400 with small blockage ratio (β = H/d, H is the height of channel) and AR = 0.5, 1 and 2. They reported that the Strouhal number remains constant for large values of Reynolds number but vortex formation region starts to be increased by increasing the values of aspect ratio. Nakagawa *et al*., (1999) performed their experimental study to find out the turbulence characteristic for flow over a rectangular rod at Re = 15000 by taking various values of aspect ratio. They examined that at AR = 2, the shear layer emerging from leading edge reattached to side walls of rod and rolled up after reaching the downstream position and appeared in the form of vortices. A numerical simulation of flow over a rectangular rod is conducted by Islam *et al*., (2012) using LBM in order to find the ramification of aspect ratio having the range $0.14 \leq \mathrm{AR} \leq 4$ and different Reynolds number $(100 \leq \mathrm{Re} \leq 250)$.They studied force coefficients, suppression of vortex shedding and flow behavior around and behind the rod. A drastic change in force coefficients and vortex shedding was observed within the range $0.15 < \mathrm{AR} < 2$, also the reduction in mean drag force was found with increment in aspect ratios from 0.15 to 2. The numerical study based on forced convection flow over a rectangular rod at Re = 100 and 200 and aspect ratio having the values 1, 2 and 3 was done by Joda *et al*., (2008). The authors used fractional step finite volume code and concluded that mean drag coefficient reduced with increment in aspect ratio for Re = 100, but for Re = 200, firstly showing increasing and then decreasing behavior with respect to aspect ratio. In case of rectangular rods, the variation in aspect ratio alters the fluid dynamic behind the rods and for small AR, the region of vortex formation is smaller in case of rectangular rod as compared to square rod. When one object is placed in the near wake of other one, the behavior of the flow and fluid forces depend strongly on the structure of an object, gap between the objects, arrangement of the objects, and wind direction.

It is therefore essential to study these characteristics practically, that may be connected to damaging of structural. The flow structure mechanism for more than one rods is quite different from a single rod. Many studies for flow over two rods in tandem arrangements are available in literature. Tandem structure plays vital role in many applications of engineering such as heat exchangers, cooling equipment, cabling in sea, wind engineering etc. These structures suffer with fluid forces when air or water interact them and could be damaged. In order to reduce these forces, many experimental and numerical investigations are conducted under the effects of some physical parameters such as Re, g and AR. Xu and Zhou (2004) performed an experiment to measure Strouhal number in region of two tandem circular rods by taking Re from 800 to 4.2×10^4 and gap spacing from 1 to 15. They examined that g = 3.5 to 5 is optimum gap spacing. An experimental study performed on flow between two tandem circular rod by Zhou and Yiu (2006) to find out the momentum, heat transport between two rod and flow structure. The chosen spacing ratio lie in the range 1.5 to 6 at Re = 7000. They investigated significant reduction in mean drag force and vortex shedding suppression. Han *et al*., (2012) numerically investigated flow behavior over two circular rods arranged in tandem for g = 1.2 to 10 at Re = 200 using the spectral element method. They obtained a critical value of g is 3.6 at which the transition in flow mode occurs due to generation of vortices from both the rods. They also examined that the drag force of both rods is less than drag force of single rod. Two dimensional numerical simulations have been carried out by Patil*et al.*, (2009) to examine the effect of size of rod and spacing ratios between two square rods. It was concluded that vortex shedding besides the downstream rod suppressed for large size of rod.

The drag force for upstream rod remains positive but the drag force for downstream rod becomes more negative by increment in the size of rods. An experimental study to measure the flow field near two square rods is done by Moon et al., (2008) using the particle image velocimetry (PIV). The spacing ratio between the rods was ranging from g = 0.5-10 at Re = 5300 - 16000. It was found that the flow modes at g = 2 are quite different from that of g = 2.5 for both selected Reynolds number. The numerical simulations are performed to analyze the flow transitions from steady to unsteady state by Abbassi and Islam (2018) for flow over two inline square rods. The Reynolds number is taken from 1-110 at fixed g = 3.5. Three different states of flow were observed, that are steady, quasi steady and unsteady states. A significant reduction in mean drag force was found as compared to single square rod. Kim *et al*., (2008) conducted an experimental measurements to find out the effect of high values of Re on the flow over two inline square rods and reported that the flow modes suddenly changed due to reattachment of the shear layers generated from upstream rod. Some other studies based on flow over two rod to examine its characteristics can be found in [Sakamoto et al., 1987; Sohankar, 2011; Etminan, 2013).

This paper is organized as; numerical method with problem description and boundary conditions is described in Section 2. The code validation and computational domain study is presented in Sections 3. The obtained numerical results are systematically shown in Section 4, focusing on the influences of Re = 80-200 and g

= 0.5-6 on flow structure mechanism. Finally, conclusions are presented in Section 5.

Problem Formulation and Numerical Method

The schematic flow configuration and details of numerical method used for flow over two rectangular rod for different gap spacings at various Reynolds number are discussed in this section.

Problem Formulation

In schematic flow configuration two rectangular rods with length d and width w are placed in horizontal direction as shown in Figure 1. The channel length is L and height is H. An upstream distance of L_u = 6d and downstream distance of L_d = 25d has been selected. In Figure 1, C1 and C2 represent upstream and downstream rods, respectively. The computational domain for proposed problem is fixed in transverse direction (y) and changes in longitudinal direction (x) for different gap spacings having the range g = 0.5 – 6 and Re = 80 – 200 (see Table 1). Mean flow velocity U_∞ is used at the inlet using the equilibrium particle distribution function where (Sukop and Thorne, 2007)

$$u = U_\infty \text{ and } v = 0 \qquad (1)$$

The convective boundary condition is incorporated at the outlet boundary (Cheng *et al*., 2007). A no-slip boundary condition is used on the surfaces of both rods as well top and bottom walls of the channel (Ziegler, 1993; Dazh., 2003).

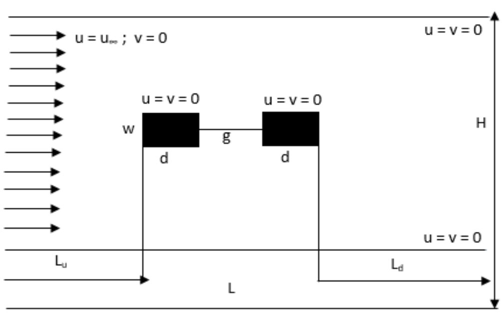


Figure1. *Schematic flow configuration for flow over two rectangular rod*

Table1. *Computational domain for different selected cases*

Cases	L×H	Cases	L×H
g = 0.5	671×221	g = 3	711×221
g = 1	681×221	g = 4	741×221
g = 1.5	691×221	g = 5	761×221
g = 2	701×221	g = 6	781×221

Lattice Boltzmann Method

The Lattice Boltzmann method (LBM) is a numerical technique used to solve models for fluid flows to find position and velocity of particles at each time step (Figure 1). It is based on two process (i) streaming and (ii) collision. Both process are local in nature which provide parallel computing (Mohammad, 2011). LBM is conditionally stable method and depends on relaxation parameter τ (Wolf-Gladrow, 2000). Moreover, the pressure can be obtained by solving the equation of state in this model [Mohammad, 2011; Wolf-Gladrow, 2000]. The evolution equation of the fluid particles can be described

$$f_i(\mathbf{x} + \mathbf{e}_i, t + 1) = f_i(\mathbf{x}, t) - [f_i(\mathbf{x}, t) - f_i^{(eq)}(\mathbf{x}, t)]/\tau \quad (1)$$

Where, f_i is the particle density distribution function, $f_i^{(eq)}$ is the equilibrium distribution function at position $\mathbf{x}$ and time t, $\mathbf{e}_i$ is the velocity vector and τ is the stability parameter. The equilibrium distribution function is calculated as

$$f_i^{(eq)} = \rho w_i[1 + 3(\mathbf{e}_i.\mathbf{u}) + 4.5(e_i.\mathbf{u})^2 - 1.5\mathbf{u}^2, \quad i = 0,1,2,\dots,8 \qquad (2)$$

$\mathbf{u}$ is the mean flow velocity and w_i are the weighting coefficients (w_i = 4/9 ,1/9 and 1/36 for i = 0 - 8). The kinematic viscosity ν is calculated as

$$\nu = (2\tau - 1)/6\Delta t, \qquad (3)$$

where Δt is time step. The flow velocity $\mathbf{u}$ and density ρ can be computed as

$$\rho = \sum f_i \text{ and } \rho\mathbf{u} = \sum \mathbf{e_i} f_i \quad i = 0,1,2 \dots 8 \qquad (4)$$

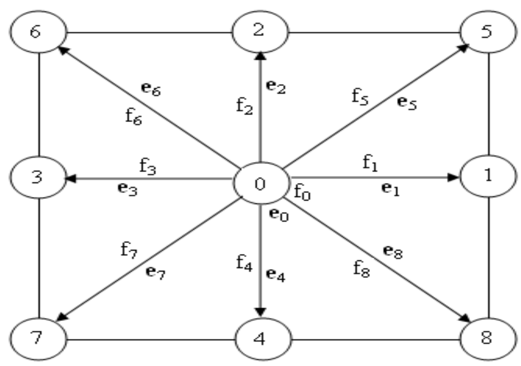


Figure2. *Nine particles distribution in two-dimensional domain*

Computational domain study and Code Validation

In order to validate the results obtained from present study, we have computed the values of fluid forces such as Cdmean and St for flow over a single square rod at Re = 100 – 200 and compared the obtained results with those available in literature, for flow over single square rod (see Table 2&3). It can be examined that the present result shows a close relationship with available experimental and numerical data for Cdmean (Cd) and St at Re = 100 - 200. Therefore, present code can captured the flow behavior in an appropriate way.

Table2. *Code validation study for flow over single square rod in terms of Cdmean*

Re/Cdmean	100	150	175	200
Present	1.41	1.42	1.44	1.48
Dutta et al. (2004) **exp**	1.15	…….	1.43	1.41
sohankar (1995) **num**	1.444	1.408	………	1.424
De and Dalal (2006) **num**	1.41	1.3982	1.412	1.3842
Gera et al. (2010) **num**	1.461	1.411	………	1.487
Cheng et al. [25] **num**	1.44	………	1.472	1.45

Table3. *Code validation study for flow over single square rod interms of St*

Re/St	100	150	175	200
Present	0.146	0.154	0.156	0.155
Dutta et al. (2004) **exp**	0.126	……..	……..	0.154
sohankar (1995) **num**	0.145	0.161	0.165	0.165
De and Dalal (2006) **num**	……..	……..	………	……..
Gera et al. (2010) **num**	0.129	0.141	0.143	0.143
Cheng et al. [25] **num**	0.144	……….	………	0.152

In order to select an appropriate grid points for suitable results and computational time, we have calculated the values of force statistics for the flow over two in-line square rods at different values of upstream (L_u), downstream (L_d) and height of the channel (H). A comparison of these values is shown in Table. 3. It can be examined that at fixed values ofL_d and H at 25d and 11d, respectively, the Cdmean and Cdrms of both the rods decreases by increasing the values of upstream distance (L_u). Therefore, we have selectedL_u = 6d in present simulation. The better results can also obtained by taking L_u = 8d but it required more time due to more grid points as compared to L_u = 6d. Similarly at fixedvalue of Lu at 6d and taking the different values of L_d and H, it can be examined that at L_d = 20d and 30d with H = 11d, the results approximately matched with those results that are obtained from L_u = 6d; L_d = 25d; H = 11d. Therefore, we have takenL_u = 6d, L_d = 25d and H = 11d for present study to obtain accurate results in less computational time.

Table4. *Effect of computational domain at Re = 100 and g = 3*

Cases	Cd1	Cd2	Cdrms1	Cdrms2
L_u=4d; L_d 25d ; H=11d	0.636	0.136	0.00011	0
L_u =6d; L_d 25d ; H=11d	0.619	0.134	0.00001	0.00002
L_u =8d; L_d 25d; H=11d	0.617	0.134	0.00001	0
L_u =6d; L_d 20d; H=11d	0.619	0.134	0.00001	0.00002
L_u =6d; L_d 30d; H=11d	0.619	0.134	0.00001	0
L_u =6d; L_d 25d ; H=9d	0.634	0.147	0.00004	0
L_u =6d; L_d 25d; H=13d	0.612	0.134	0.00001	0.00001

Table5. *Percentage reduction in Cdmean and Cdrms of two rods at Re = 100 and g = 3*

Single rod		Two rods	
		Cdmean1	Cdmean2
Cdmean	0.6369	0.6188	0.1340
% reduction	---	3.8%	24.43%
Cdrms	0.0000255	0.00000928	0.00001214
% reduction	….	63%	52.4%

To study the effect of two rectangular rods instead of single one in stream wise direction, on the force coefficients, we have computed the values of Cd mean and Cd rms of single and two inline rectangular rods A comparison of these values with percentage reduction is shown in Table. 5. It can be seen that the Cd mean of single rod is highest as compared to two rods and percentage reduction in Cd mean of C2 is maximum than C1, while root mean square values of drag coefficient for C1 is less than C2.

From here it can be inferred that for more than one object in inline arrangement, the value of mean drag coefficient decreases due to effect of upstream rod on downstream rods. Therefore, in present work, we have chosen two rectangular rod instead of single one.

RESULTS AND DISCUSSIONS

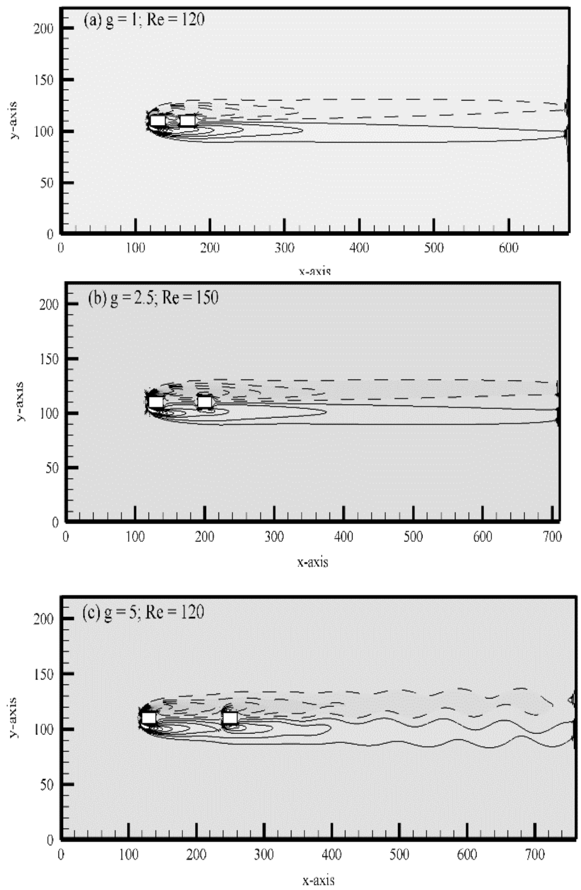
A present 2-D study is based on to analyze the behavior of fluid flow over two inline rectangular rods by varying gap spacings between the rods as g = 0.5-6 and Re = 80-200. The results are obtained in terms of vorticity contour visualization, power spectrum analysis and force statistics. For sake of simplicity and to avoid the repetition of results, only some important representative plots will be shown over here.

Vorticity Contour Visualization and Power Spectrum Analysis

This section based on vorticity contours visualization for different flow behavior and energy spectra analysis of lift coefficients for both rectangular rods. Under the effect of gap spacing (g = 0.5 – 6) at different Reynolds numbers having the range Re = 80 – 200, four different types of flow modes are found that are named as (i) Steady flow mode, (ii) Quasi steady flow mode, (iii) Fully generated single slender body flow mode and (iv) Fully generated irregular vortex shedding flow mode. First flow mode is obtained for small spacing such as g = 0.5–3 for all selected Reynolds number i.e Re = 80–200. In this flow mode, flow represent study behavior throughout the channel, due to dominancy of viscous forces, that resist the movement of flow, no matter how large is the value of Reynolds number (see Fig 3(a, b)). This flow mode is also observed at (g, Re) = (4, 80), (4, 100), (5, 80), (5, 100), (6, 80), (6, 100) and (6, 120). In that cases, no significance effect of Reynolds number on flow behavior is found at various gap spacing and flow moves smoothly in whole channel. Since in steady flow mode, no vortices seemed to be appeared within the gap as well downstream position. Therefore, we can’t compute the values of Strouhal number because of steady state of the flow and its graph shows the straight line passed through origin. Steady flow mode also observed by Harichandan and Roy (2010) at Re = 100 and Abbassi and Islam (2018)at g = 3.5, Re = 54 for two square rods.

Next when Re is 120 for g = 5, a small disturbance in flow is observed due to its rotational motion. The symmetric behavior of flow is observed within the gap of both rods and vortices start to be appeared from middle of channel and propagate towards the exit of channel, but proper vortex shedding is not formed. The positive vortices start to be generated from bottom corner and negative vortices will be formed from top corner of downstream rod, respectively (see Fig. 3(c)). This flow mode is called quasi steady or nearly unsteady (QS) flow mode and observed only for the case of (g, Re) = (5, 120). Energy spectrum graphs for QS flow mode showed single broad banded peak due to wide wake region. In comparison of magnitude of spectrum energy for both the rods, downstream rod attains higher magnitude of spectrum energy as compared to upstream one. Because rotational motion of flow starts from top and bottom corners of downstream rod instead of upstream one (see Fig 4(a, b).

For medium or large value of g and Re,flow is no more steady or quasi steady.The onset of vortex shedding is started for g > 3 and Re > 120. Therefore,third flow mode based on unsteady behavior of flow and named as Fully generated single slender body(FDSSB). It is observed when the gap spacing is taken as g = 4 and Re is varied from Re = 120–200. The vortices seemed to be completely appeared within the gap due to enough space and propagated towards the downstream position in an alternate style after passing over the second rectangular rod. It is noticed that the transformed vortices at downstream side are little bit oval shape due to high frequency of fluid flow on the second rod (see Fig 3(d, e)) and move towards the exit position as a single slender body. That’s why we called this flow mode fully generated single slender body flow mode. Similar flow mode is visualized for (g, Re) = (5, 150) and (5, 175) and (6, 150). Which cleared that for large gap and high values of Reynolds number play prominent role for disturbing steady state of flow by the formation of vortices and produced more fluid forces. This flow mode was also found by Sohankar(2011) at g ≥ 4 and Re = 160 for flow over two tandem square rods Spectrum energy for FDSSB flow mode gives the single sharp peak for all cases but the frequency magnitude of downstream rod is greater than upstream rod. Primary frequency is observed dominantly while secondary frequency is negligible due to regular generation of vortices at downstream location shown in Fig 4 (c, d). The last existing flow mode is observed at maximum values of Reynolds number and gap spacing i.e (g, Re) = (5, 200), (6, 175) and (6, 200) and named as FDIVS flow mode. In this flow mode, due to enough spacing and maximum value of Re, the vortices fully formed within the gap of upstream and downstream rectangular rods, when these generated vortices hit the downstream rod, they propagate as a irregular manner towards the downstream location. No proper behavior of propagation and shape of vortices is noticed because of high frequency of the fluid forces. In that flow mode the formation length of vortices behind the downstream rod becomes longer as compared to FDSSB flow mode (see Fig 4). The graph of Spectrum energy for FDIVS flow mode is presented in Fig 4(i-l). Single sharp peak is appeared for both selected cases related to FDIVS flow mode. The magnitude of spectrum energy of C2 is greater than C1.It is observed here that at (g, Re) = (5, 200) greater value of energy spectrum is found than at (g, Re) = (6, 175) (see Fig 4 (i-l)).



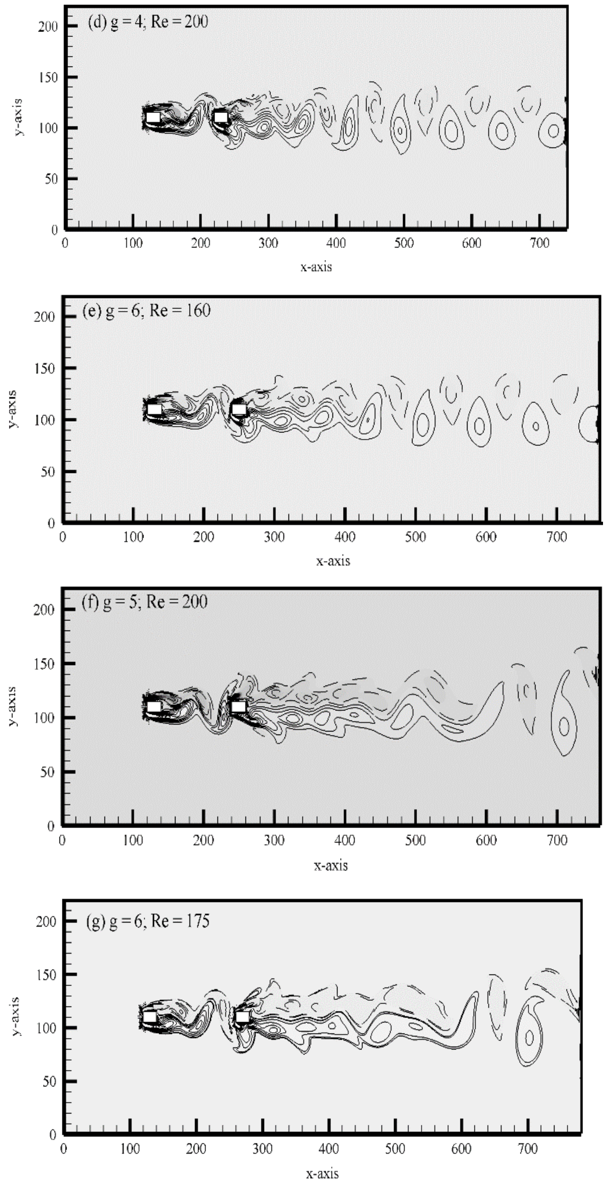
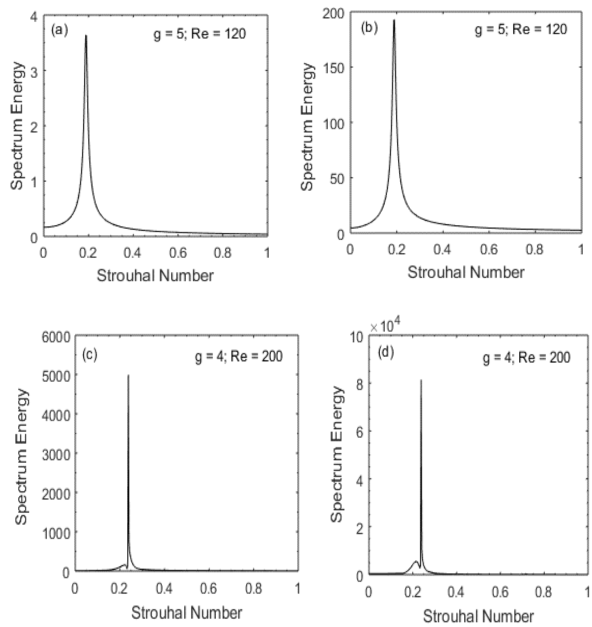


Figure3. *Vorticity contours for different flow modes*



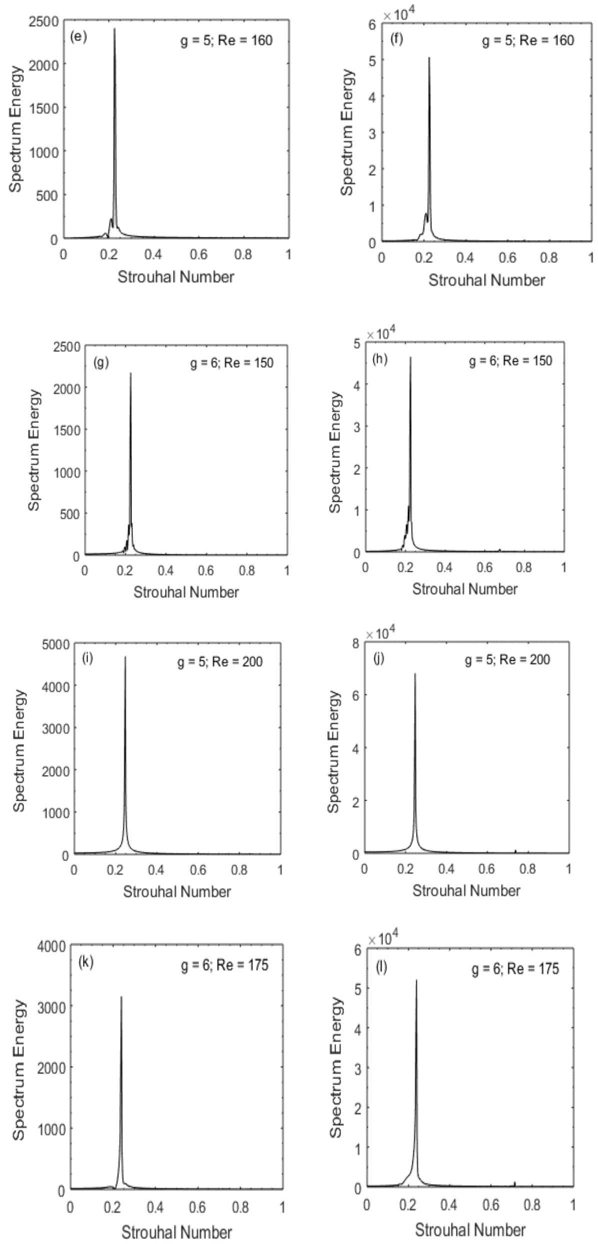
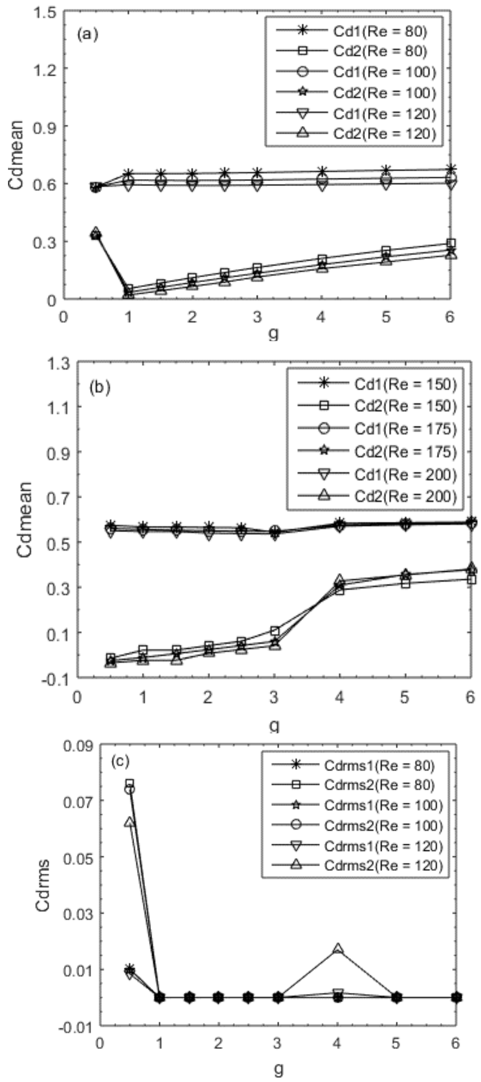


Figure4. *Spectrum Energy of lift coefficients for different flow modes*

Force Statistics

The effect of gap spacing between two tandem rectangular cylinders at various Rein terms of force statistics, such as mean of drag coefficient (Cdmeam), root mean square of drag coefficient (Cdrms), root mean square of lift coefficient (Clrms) and Strouhal number (St)is presented in Fig. 5(a-h). The obtained values are compared with each other to find critical gap spacing and Reynolds number. The Cdmean of up and downstream rods is presented in Fig.4 (a, b) against gap spacing i.e g = 0.5–6 at fixed Reynolds number (Re= 80–200). It is noticed that mean drag coefficient of upstream rod is greater than the downstream rod for all selected Reynolds number. Its values are decreasing from g = 0.5 to 1 at Re = 80 – 120, but after that an increasing behavior is visualized with increment in gap spacing. For Re = 150 – 200, Cdmean values decrease from g = 0.5 – 3, after that it start to be increased with increment in gap spacing values from g = 4 – 6. The maximum value of Cdmeanis attained by C1 at (g, Re) = (6, 80),that is 0.6734, where Steady flow mode is observed. The downstream rod having an increasing trend with gap spacing except for g = 0.5 to 1 at Re = 80 – 120 andattained its maximum value at (g, Re) = (6, 200), that is 0.3807, where FDIVS flow mode is characterized. While the minimum value of C2 is occurred at (g, Re) =(0.5, 200), which is -0.0348. The effect of thrust is also observed for second rod at Re ≥150 for g = 0.5–1.5 for steady flow mode. The reason of thrust is that at small gap and high Re both drag and thrust forces affect each other's impacts on both cylinders and first cylinder exerts the pressure on downstream rod but as the gap increased, the effect of thrust vanishes.



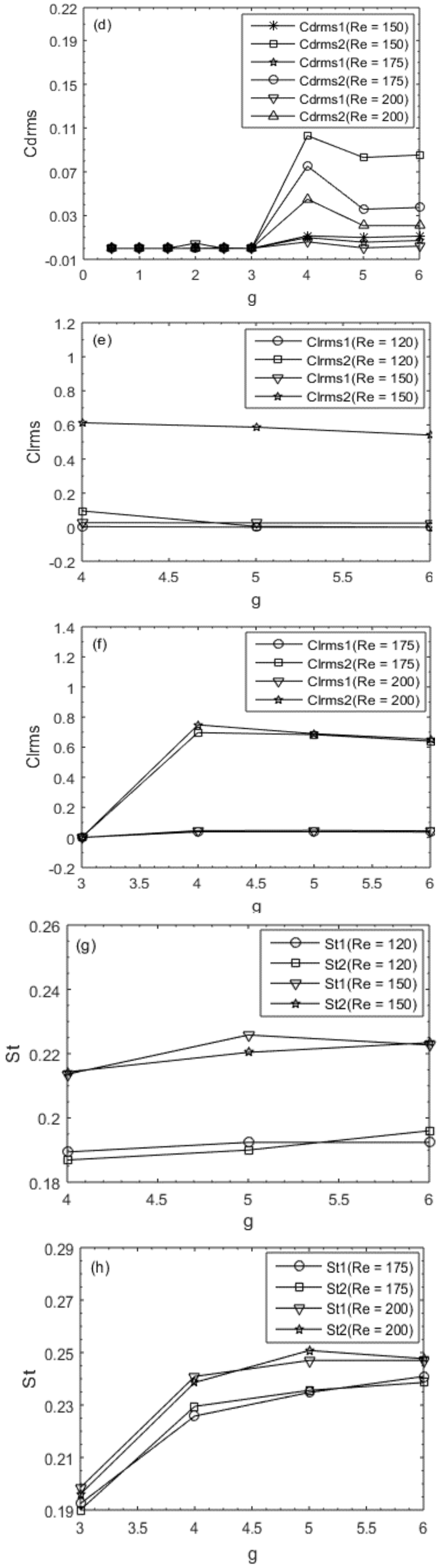


Figure5. *Variation of (a, b) Cdmean, (c, d) Cdrms, (e, f) Clrms and (g, h) St.*

The root mean square values of drag coefficients for both cylinders are presented in Fig. 5(c, d). From Fig. 5(c) it is cleared that Cdrms of downstream rod attained the maximum values at g = 0.5 for Re = 80–120 than upstream one. But when gap is considered as g = 1–6 for Re = 80 – 120, Cdrms of C1 and C2approximately attained the same values except at g = 4 and Re = 120 for C2. The Cdrms value of downstream rod again shoots up at g = 4 for Re= 150–200 and attained some maximum value with visible increasing and decreasing mode (see Fig. 5(d)). The maximum value of Cdrms for upstream and downstream rod is occurred at(g, Re) = (3, 150)that is 0.1953 and 0.2133, respectively. Where existing flow mode is steady. The minimum values of both up and downstream rods are obtained at (g, Re) = (2, 80) that are 0.000001898 and 0.000001666, respectively (see Fig. 5(c)).

The Clrms values and Strouhal number are presented only for g ≥ 3 at Re ≥ 120 in Fig. 5(e-h), because at small values of g and Re, flow remains steady and no vortex shedding seemed to be observed throughout the channel. Therefore, at that values of g and Re, Clrms values and St are not computed. The root mean square value of lift coefficient for C2 is greater than C1 for all selected Reynolds number. It shows decreasing behavior with increment in g except at Re = 200, where Clrms value first increases from g = 3 to 4 and after that it decreases. At (g, Re) = (4, 200),Clrmsattains maximum value for C2 i.e0.7484, where FDIVS flow mode is obtained. The Strouhal number against gap spacing at fixed Reynolds number is drawn in Fig. 5(g, h). Its values having increasing trend with increment in gap spacing for Re = 120 – 200. In contrast to Cdmean and Clrms, C1 attains larger values of St than C2.The maximum value of St is 0.2508 acquired at (g, Re) = (5, 200), where flow is fully developed within the gap as well downstream location of second rod.

Conclusions

A (2-D) numerical study for the unsteady laminar flow past over two inline rectangular cylinders is carried out by Lattice Boltzmann method to study the effect of Reynolds number and gap spacing between the rectangular rods. The Reynolds numbers are taken within the range 80 – 200 with gap spacing ranging from 0.5 – 6. The main findings are:

- Four various types of flow modes are examined, named as: steady flow, Quasi-steady flow, fully generated single body and fully generated irregular vortex shedding flow modes.
- The steady flow mode is obtained for $0 \leq g \leq 3$ at $80 \leq Re \leq 200$ and $4 \leq g \leq 6$ at $80 \leq Re \leq 100$.
- Unsteady flow behavior is observed for larger values of Reynolds number ($Re > 120$) and gap spacing ($g > 3$), where existing flow modes are fully developed single slender body and fully developed irregular vortex shedding.
- The mean drag force for upstream rod (C1) is higher than downstream rod (C2) for all selected gap spacing at fixed Reynolds number. (v) The maximum value of Cdmean is 0.6734 obtained at (g, Re) = (6, 80).
- Some negatives values of Cdmean for second rod are observed at g = 0.5 for Re > 120, g = 1 for Re > 150 and g = 1.5 for Re = 200, respectively. These negative Cdmean values show the effect of thrust.
- The root mean square values of lift coefficients for downstream rod is greater than upstream rod, because of strong generation of vortex shedding from C2.
- The maximum value of Strouhal number is acquired from C2 at (g, Re) = (5, 200) and that is 0.2508.

REFERENCES

[1] Dutta S, Panigrahi PK, Muralidhar K. Effect of orientation on the wake of a square cylinder at low Reynolds numbers. Indian J Eng Mater 2004:11:447–459.

[2] Gera B, Sharma PK, Singh RK. CFD analysis of 2D unsteady flow around a square cylinder. Int J ApplEng Res. 2010:1:602–610.

[3] Sohankar A, Davidson L, Norberg C. Numerical simulation of unsteady flow around a square two-dimensional cylinder. In: Twelfth Australasian Fluid Mechanics Conference, The University of Sydney, Australia 1995:517–520.

[4] Okajima A. Strouhal numbers of rectangular cylinders. J. Fluid Mech, Digital Archive, Cambridge University press 2006:123:379-398.

[5] Bearman PW and Trueman DM. An investigation of the flow around rectangular cylinders. Aeronant Quart 1972:23:229-237.

[6] Shadaram A, Fard MA and Rostamy N. Experimental study of near wake flow behind a rectangular cylinder. Amer. J. Appl. Sci 1980:5(8):917-926.

[7] Nakagawa S, Nitta K andSenda M. An experimental study on unsteady turbulent near wake of a rectangular cylinder in channel flow. Exper. Fluids 1999:27(3):284-294.

[8] Islam SU, Zhou CY, Shah A and Xie P. Numerical simulation of flow pastrectangular cylinders with different aspect ratios using the incompressible lattice Boltzmann method. J. Mech. Sci. Tech 2012:26 (4):1027-1041.

[9] JodaA, Cuesta I and Vernet A. Numerical study of forced convection flow around rectangular cylinders with different side ratios. Thermal issues in Emegring Technologies, THETA 1, Cairo, Egypt 2008.

[10] Xu G and Zhou Y. Strouhal numbers in the wake of two inline cylinders. Exp. Fluids 2004:37:248-256.

[11] Zhou Y and Yiu MW. Flow structure, momentum and heat transport in a two-tandem-cylinder wake. J. Fluid. Mech 2006:548:17-48.

[12] Han Z, Zhou D and Gui X. Flow past two tandem circular cylinders using Spectral element method. The Seventh International Colloquium on Bluff Body Aerodynamics and Applications (BBAA7), Shanghai, China 2012.

[13] Patil PP, Tiwari S. Numerical investigation of laminar unsteady wakes behind two inline square cylinders confined in a channel. EngApplComput Fluid Mech2009:3(3):369–385

[14] Moon KK, Dong KK, Soon HY and Dae HL. Measurements of the flow fields around two square cylinders in a tandem arrangement J MechSciTechnol2008:22:397-407.

[15] Abbassi WS, Islam SU. Transition from steady to unsteady state flow around two inline cylinders under the effect of Reynolds numbers. Journal of the Brazilian Society of Mechanical Sciences and Engineering 2018 :40:168.

[16] Kim MK, Kim DK, Yoon SH, Lee DH. Measurements of the flow fields around two square cylinders in a tandem arrangement. J MechSciTechnol2008:22:397–407.

[17] Sakamoto H, Haniu H, Obata Y. Fluctuating forces acting on two square prisms in a tandem arrangement. J Wind Eng Ind Aerodyn 1987 : 26:85–103.

[18] Sohankar A. A numerical investigation of the flow over a pair of identical square cylinders in a tandem arrangement. Int J Numer Methods Fluids 2011:70:1244–1257.

[19] Etminan A. Flow and heat transfer over two bluff bodies from very low to high Reynolds numbers in the laminar and turbulent flow regimes. Int J Adv Des Manuf Technol 2013:6(2):61–72

[20] Mohammad AA. Lattice Boltzmann Method: Fundamentals and Engineering Applications with Computer Codes, Springer 2011.

[21] Wolf-Gladrow DA. Lattice-Gas Cellular Automata and Lattice Boltzmann Models: An Introduction, Springer-Verlag Berlin Heidelberg 2000.

[22] Cheng M, Whyte DS and Lou J. Numerical simulation of flow around a square cylinder in uniform-shear flow.J Fluid Struct 2007: 2(23): 207-226.

[23] Ziegler. Boundary conditions for lattice Boltzmann simulations.J. Stat Phy 1993:71:1171-1177.

[24] Sukop MC and Thorne DT.Lattice Boltzmann Modeling: An Introduction for Geoscientists and Engineers, Springer 2007.

[25] Dazhi Y,Renwei M, Luo LS and Wei S. Viscous flow computations with the method of lattice Boltzmann equation.Prog Aerospace Sci2003:39:329-367.

[26] Harichandan AB, Roy A. Numerical investigation of low Reynolds number flow past two and three circular cylinders using unstructured grid CFR scheme. Int J Heat Fluid Flow 2010: 31:154-171.

[27] Sohankar A. A numerical investigation of the flow over a pair of identical square cylinders in a tandem arrangement. Int J Numer Methods Fluids 2011:70(10):1244-1257, 2011.

Citation: *Galal Ali Hassaan, "Mechanical Engineering in Ancient Egypt, Part 76: Alabaster Products Inscription", International Journal of Emerging Engineering Research and Technology, 6(10), pp.24-33*

Copyright: *© 2018 Galal Ali Hassaan. This is an open-access article distributed under the terms of the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original author and source are credited.*