Numerical investigation of flow field over four cylinders in square arrangement for different spacing ratios

P.Goutam and D Siva Krishna Reddy

Department of Mechanical Engineering, SRM Institute of Science and Technology, Kattankulathur, Chennai, India. 603203
E-mail: sivakriv@srmist.edu.in

Abstract. The objective of this paper is to study the effect of spacing ratio on the wake dynamics in an in-line square array of four cylinders. Two dimensional incompressible CFD simulations are done with commercial CFD software Ansys- Fluent. Diameter of cylinder is 0.01 m, and the four cylinders have identical diameter. The spacing ratios considered are 1.5, 2.0, 2.5 and 3.0, where spacing ratio is defined as ratio of distance between center of two cylinders and diameter of cylinder. Simulations are performed at a Reynolds number (Re) of 200. Water is considered as fluid medium, and velocity of water corresponding to Re of 200 is 0.0178 m/s. SIMPLE algorithm is used to solve incompressible Navier-Stokes equations, and second order accuracy is used for spatial discretization. At the chosen Re, far wake is expected to transition to turbulence, and hence κ-ω SST turbulence model is used for turbulence closure. The flow field and associated surface properties such as drag and lift force are strong function of spacing ratio. The impingement of shear layers of upstream cylinders onto to the downstream cylinders is strongly influenced by spacing ratio. The impingement and resulting flow interaction leads to asymmetry in the surface pressure distribution. This leads to generation of lift force on the cylinders. For spacing ratio of 1.5 and 2.0, the top row cylinders have positive lift coefficient, and bottom row cylinders have negative lift coefficient. For spacing ratio of 2.5 upstream cylinders have positive lift coefficient, and downstream cylinders have negative lift coefficient. All four cylinders have positive lift coefficient for spacing ratio of 3.0. Drag coefficient found to be positive for all cylinders and for all spacing ratios.

1. Introduction

Bluff bodies are subjected to vortex induced vibration (VIV) when they are in fluid medium either air or water or in any fluid medium VIV is a phenomenon were in the flow gets separated from surface of bluff body due to adverse pressure gradients. These separated flows referred to as shear layers roll up into vortices in the wake of bluff body and finally detaches or sheds from the shear layer in to the downstream wake. These shed vortices will impart a backward force on the bluff body which sets it into vibration. Some examples for VIV: riser tubes, mooring lines, electrical power supply lines (a unique class of VIV problem known as Galloping). Traditionally these VIV are found to cause devastating effect on the...
structures under its influence. Bridges, heat exchanger tubes are classic examples to note. Researches nowadays are harness these vibrations to generate electricity Hobbs et.al [1] because of low cost of production, even if their power wattage is in micro and milli watts and this can be used for powering electronic devices which run on small power input.

Array of cylinders which is a very common in most of the real-world applications say in riser tubes in marine applications, heat exchanger tubes have motivated researchers to study vortex induced vibration in multiple array configuration. Vortices are of different forms viz S type, P type, P+S type, 2S type, 2P type to mention a few types Williamson and Govardhan [2]. These vortices are found at different amplitudes of response from the bluff body Vortex shedding can be improved by the oscillations of cylinder as the span wise correlation is increased as noted by Bearman [3], Reynolds number the flow has as significant effect on the vortex formation in the flow field researcher like Sadatoshi Taneda (1956) [4] carried out experiments with different Reynolds number, he observed that vortices are formed as Reynolds number is increased. Wake dynamics of the array of cylinders at different spacing ratio both in tandem and side by side arrangement was studied by Zdravkovich [5]. It was observed that different wake patterns at different spacing when the cylinders were placed in tandem and in side by side arrangement, in his experiment he observed that at certain spacing the downstream cylinder was interfering with wake of upstream cylinder and hence the shedding of upstream was affected. Sumner et.al [6] has studied the Strouhal number variation for differed P/D ratio (pitch ratio) of two staggered cylinders of equal diameter, angle of inclination between upstream and downstream cylinders was varied from 0° to 90°. They observed reattachment and gap flows were observed at smaller and higher spacing. Vortex dynamics is complex for array of cylinders.

Tong et.al. [7] Four cylinders in an in-line square arrangement at Reynolds number of 100 to 500 at a fixed spacing of L/D = 2.0. From Reynolds number of 100 to 220, the array of four cylinders behave as a single cylinder and their wake has a single Karman street. Sayers [8] has carried out experiments on three and four equispaced cylinders of same diameter at Reynolds number of 3x10^4 in range of spacing from L/D 1.5 to 5 and at inclination angle (α) 0° to 180°. They observed that for four cylinders Strouhal number was higher than the single cylinder, this is because at all spacing array behaves as single cylinder and at higher spacing wake of cylinder array is quite complex. In the present work, simulations are performed over array of four cylinders in in-line square arrangement. The objective of the work is to investigate the effect of different spacing ratio (S/D = 1.5, 2.0, 2.5 and 3.0) on the cylinder array at Reynolds number of 200. Figure 1 shows geometry of the array of cylinders, this arrangement has two upstream cylinders named as C1 and C4, two downstream cylinders C2 and C4. The free stream conditions are: Reynolds number is 200 velocity of 0.0178 m/s. The cylinders have identical diameters and its value is 0.01 m. Computational domain size of 20 diameters in height and 40 diameters in width.

2. Simulation Methodology
ANSYS – Fluent is used for simulating the flow field over the array of cylinders. Simulations are performed on structured girds, which are generated using ANSYS Workbench meshing tool. The incompressible RANS equation are solved with pressure-based algorithm under steady state condition. κ-ω SST model is used to close the turbulence terms in RANS equation. At inlet, free stream conditions are imposed. At outlet, zero-gauge pressure is applied. No-slip boundary condition is applied on cylinder walls. Figure 1 shows the nomenclature of cylinders and flow direction. Figure 2 shows a plane view of the gird in XY plane. Fine mesh is used at the near wall region of the cylinder wall. The girds are progressively stretched towards outlet boundary so that a coarse grid can be employed. The first gird point
in the wall-normal direction is placed at a distance of 1.24e-07 mm from wall. The \( y^+ \) value in majority portion of the wall is below 5. The total number of cells in various grids are shown in Table 1.

![Figure 1](image1.png)

**Figure 1.** Schematic representation of four cylinders in square arrangement. Diameter of the cylinder is represented by 'D', and spacing between cylinders is indicated by 'S'.

![Figure 2](image2.png)

**Figure 2.** Schematic of the structured grid used in simulations: (a) Computational domain and (b) Enlarged view.

Figure 3 shows reduction in the predicted drag coefficient (\( C_d \)) as a function of the iterations for a typical simulation on logarithmic scale. There is a rapid drop in \( C_d \) from 0 to 100 iterations. In the subsequent iterations, the reduction in the \( C_d \) is slower. The \% change in \( C_d \) from 8000 to 10000 iterations is less than 0.1\%. As the \( C_d \) variation is negligible beyond 10000 iterations, solution is treated as converged.
Figure 3. The predicted drag coefficient as a function of iterations for a typical simulation for spacing ratio of 2.5.

Table 1. Drag coefficient obtained on three successively refined grids for spacing ratio of 2.5.

| Grid    | Number of cells | Coefficient of drag $C_d$ |
|---------|-----------------|---------------------------|
| Gird 1  | 37172           | 8.1566                    |
| Gird 2  | 144812          | 7.4255                    |
| Gird 3  | 233057          | 5.6781                    |

3. Validation of CFD results

To validate the CFD results we compared the pressure distribution profile for flow over a single cylinder of our simulation at Reynolds number of 200 with results of Thom [9] in figure 4 and the profile was found out to be in good agreement for most of the angles, with little variation that is likely due to the steady state simulation.

Figure 4. The Pressure distribution around the cylinder i.e. $C_p$ vs Theta: Thom [9] & Present Work.
4. Grid Convergence

To assess the grid point density to resolve flow gradients, a grid convergence study is performed for spacing ratio of 2.5. Simulations are performed by reducing and increasing the number of points by 50% from Grid 1 to Grid 2 and by 50% from Grid 2 to Grid 3. The near wall spacing is kept constant in all the three grids. In the circumferential direction, the cylinder is divided into 30 divisions, 60 divisions and 90 divisions for Grid 1, Grid 2 and Grid 3 respectively. Predicted coefficient of pressure ($C_p$) obtained from converged solutions on the grid 1, grid 2 and grid 3 are shown in Figures. For cylinder C1, the effect of grid refinement is clearly seen from theta $60^\circ$ to $320^\circ$. This region represents the separated flow. Solution obtained from grid 2 and 3 are nearly identical, whereas that of grid 1, deviates from rest of others. A similar behavior is observed on cylinder C3 also. For cylinders 2 and 4, the solution changes for entire $360$ deg as the grid is refined. As these two cylinders faces the separated from upstream cylinders, the flow field is more susceptible to grid refinement. The solution obtained from coarse grid has multiple maxima and minima. In case of medium and fine grids, these peaks are diffused, and results in a smoother variation of $C_p$. Again, the solution obtained from medium and finer grids are nearly identical. Overall, further refining the mesh from Grid 2 to Grid 3 does not yield any significant changes. Hence a grid 2 of 60 number of divisions is used for simulations which leads to lesser computation time as compared to Grid 3.

![Graphs of coefficient of pressure for different grids](image)

**Figure 5.** Effect of grid refinement on coefficient of pressure, (a) Cylinder 1, C₁, (b) Cylinder 2, C₂, (c) Cylinder 3, C₃, (d) Cylinder 4, C₄.
Cylinder 3, C₃, (d) Cylinder 4, C₄.

Figure 5 shows $C_p$ versus $\theta$ plots of cylinder C₁, cylinder C₂, cylinder C₃ and cylinder C₄ for three grids. The cylinders C₂ and C₄ are more sensitive to grid convergence because of the early flow separation that is at $\theta = 262^\circ$ and $\theta = 97.5^\circ$ for C₂ and C₄ respectively, this can be justified by the larger negative values of pressure coefficient $C_p$ as shown in the Table 2.

5. Results

5.1. Velocity Field

Figure 6 (a) and (b) shows the velocity contours for spacing ratio 1.5. The main flow features such as flow expansion around the fore stagnation point of C₁ and C₂; accelerated flow between the gaps of cylinders in the horizontal direction (which is known as gap flow), near wake in the downstream of C₂ and C₄ can be identified. The flow expansion extends up to the near wake region. Figure 6(b) shows the enlarged view of the flow filed in the neighborhood of cylinders. The stream lines are superimposed for better visualization of the flow field. The flow in the between the cylinders is marked by the presence of multiple vortices. The top shear layer from C₁ shifts away from the downstream C₂ cylinder, whereas the bottom shear layer impinges on the C₂. After impingement, a part of the flow, moves in the upward direction, which pushes the shear layer emanating from the top side. This avoids the impingement of top shear layer on the C₂. Now the remaining part of the impingement flow, moves along with the gap flow. Figure 7 shows the enlarged view near the impingement point. The flow is visualized in terms of vectors. This clearly shows the bifurcation of impingement flow into two streams. A similar behavior can be noticed for C₃. The top shear layer from the C₃ impinges onto the C₄, and pushes the bottom shear layer downward. The flow bifurcation at the impingement point on C₄ is also shown in Figure 7. The bottom shear layer from C₁, and top shear layer from C₃, compresses the gap flow, and results increase of flow velocity in the gap region. The near wake region, in downstream of cylinders also marked by presence of multiple vortices. The bottom shear layer form C₃, and the gap flow will intersect with each other at a distance of 0.0549 times diameter from the center of C₄. This point is known as confluence point. The vortices formed in the near wake region are not detaching from wake, and hence not able to shed beyond confluence point. On contrary, the top shear layer of C₁ and gap flow will not converge (see Figure 6(a)). This provides a way for the vortices detach from wake and convect along the flow. This results in the formation of vortex shedding in far wake of C₂ cylinder. It is evident from figure that wake of top row of cylinders is wider than that of the bottom row of cylinders. A similar observation was found in Ref. [5], and this flow known as biased flow. Figure 6(c) and 6(d) shows velocity contours for spacing ratio of 2. The overall pattern of flow field is similar to that of spacing ratio 1.5. But some of the local flow features differ significantly as explained below. The top and bottom shear layers of top row of cylinders’ merge, and results in formation of long slender separation bubble. Presence of multiple vortices can be noticed in the separation bubble (see Figure.6 (c) and 6(d)). The formation length is relatively higher than that observed for spacing ratio of 1.5. In a similar way, the top and bottom shear layers from bottom row of cylinders merge and form a recirculation bubble. Presence of multiple vortices can be noticed in this region also. The width of near-wake behind the top row and bottom row of cylinders is nearly same. This is in contrast to that of 1.5 spacing ratio.
Figure 6. Numerical visualization of the flow-field in terms of the velocity contours for Re of 200: (a), (c), (e), (g) represent entire computational domain, and (b), (d), (f), (h) depicts magnified view near the cylinders. Streamlines are superimposed on velocity contours.
The gap flow approximately remains uniform until it crosses the downstream cylinder. Downstream of the cylinders, there is exchange of momentum between the gap flow and wakes of top and bottom cylinders. This results in decrease of velocity of gap flow, and leads to formation of second set of shear layers. The impingement point on the top row cylinder shifts downwards as compared to that of 1.5 spacing. The bifurcated flow from the impingement point will move towards the upstream cylinder, and not able to merge with the top shear layer. This results in the less interference effects in the in-between region of cylinders. A similar pattern can be observed for bottom row of cylinders. Overall, the above pattern of flow field results in vortex shedding in the downstream of near-wake region of both top row and bottom row of cylinders. The velocity contours for spacing ratio 2.5 are shown in Figure 6(e) and 6(f). The flow pattern is entirely different than that observed for above two cases. The expansion of the flow over top and bottom row of cylinders is irregular. Downstream of C1 and C3, vortices are formed, and size of the vortices are nearly as same as diameter of the cylinder. Formation length of cylinders are shorter than the earlier spacing of 1.5, 2.0. The top shear layer of C1, impinges on the C2. A part of impingement flow moves away from the C2, in positive y direction. The other part of the impingement flow moves downward, mixes with downward shear layer of C2. This merged flow roll up and pushes the top shear layer. The resulting flow interaction leads to roll up of gap flow also. A similar behavior is observed in the bottom row of cylinders.

As the formation length is shorter, the flow is more prone to vortex shedding. The vortices attain sufficient circulation in the near wake region, and cut off themselves from the separation bubble. The vortex shedding occurs from both top and bottom row of cylinders. At the spacing ratio of 2.5, the cylinders behave individually rather than as a unit of array of cylinders. Due to above reasons, spacing ratio of 2.5, termed as critical spacing. The overall flow pattern for spacing ratio of 3 (see Figure 6(g) and 6(h)) is similar to that of spacing ratio of 2.5. Top shear layer of C1 deflects downward and lifts up again to move over the front side of C2. The upper portion of the gap flow, will sneak into the space between the top row of cylinders, and moves over the front side of the C2. This results in the merging of upper portion of the gap flow and deflected flow from the top shear layer of C1. The momentum of deflected flow is not good enough push the gap flow downward, and hence the deflected flow will move along with the gap flow. This is in contrast to the 2.5 spacing ratio, where the deflected flow moved in the downward direction. Shear layers from the C2 merge each other at a location, immediately downstream of the cylinder. However, the shear layers are not symmetric and shifts downward. The formation length for top row...
row of cylinder also lesser. The resulting wake interaction results in onset of vortex shedding. The flow over bottom row of cylinder is similar to that of top row of cylinders. However, a strong vortex is noticed in the downstream of C3. The formation length of the downstream cylinder is higher compared to that of top row of cylinders. Vortex shedding occurs in the downstream of confluence point.

5.2. Variation of Cp with respect to spacing ratio

**Figure 8.** Cp variation with respect to angle for (a) cylinder C1 (b) cylinder C2 (c) cylinder C3 and (d) cylinder C4.

Coefficient of pressure (Cp) for C1 is presented in Figure 8(a). As expected, at the nose stagnation point (theta =0), Cp value obtained from all solutions is nearly same. This indicates the Cp value at nose stagnation is not influenced by the spacing between the cylinders. As distance from the nose stagnation point is increased, the curves deviates from each other. The deviation is more prominent after 60 deg. The flow separation region is marked by the low values of Cp, and it spans approximately from 60 deg to 270 deg. In the separation region, Cp values obtained from spacing ratio of 1.5 and 2 are nearly same and remains constant. Beyond 270 deg (negative Y axis), in both cases, Cp increases monotonically up to nose stagnation point. However, the deviation between the two curves is much higher than that is noticed from theta 0 to 60. This is essentially due to the impingement of bottom shear layer of C1 onto the C2, and
subsequent flow interaction. The trends observed for S/D of 2 and 2.5 in the attached boundary layer region are similar to those observed for S/D of 1 and 1.5. However, they differ significantly in the separation region. There is a non-monotonic increase in $C_p$. This is due to the presence of larger vortex in the space between the two cylinders. Figure 8 (b) shows the $C_p$ variation on C2. The $C_p$ values over the entire circumference of the cylinder are lower compared to those observed for C1. This is due to the fact that, the cylinder C2 placed in the wake of C1. So C2 is exposed to separated flow in its entire circumference. $C_p$ variation between S/D of 1.5 and 2 are similar. They follow similar behavior although the magnitudes are different. In contrast, trends observed for C3 and C4 are different. For these two cases, nose stagnation point do not occur at $\theta = 0$. As explained earlier, this is due to the impingement of upper shear layer of C1 on the C2. The resulting flow interaction results in irregular $C_p$ variation. The trends observed on C3 and C4 (see Figure 8 (c) and 8 (d)) are qualitatively similar to those for C1 and C2 (see Figure 8(a) and 8(b)) respectively.

5.3. Comparison of Drag and Lift Coefficients

Comparison of drag and lift coefficients for different spacing ratio is presented in Table 2. For spacing ratio 1.5, drag coefficient for C1 and C3 is about 1.9; for C2 and C4 is about 0.1. On upstream side, the C1 and C2 are exposed to free stream flow, whereas on the downstream side, the cylinders are being surrounded by separated flow. This results in a huge pressure difference upstream and downstream side, resulting higher $Cd$ values. On the other hand, C2 and C4 are in the wake of upstream cylinders, and are impacted by a lesser velocity flow. This results in lesser $Cd$ value. For spacing ratio of 2, a similar behavior can be noticed. For spacing ratio of 2.5 and 3, the drag coefficient for C1 and C2 are comparable to each other. As explained earlier, this is mainly due to the increased velocity of the flow, upstream of the C2. Similarly, $Cd$ of C3 and C4 are comparable.

| Spacing ratio | Cylinder | Coefficient of drag $C_d$ | Coefficient of lift $C_l$ |
|---------------|---------|---------------------------|--------------------------|
| 1.5           | C1      | 1.9546                    | 0.47008                  |
|               | C2      | 0.12256                   | 0.11159                  |
|               | C3      | 1.9682                    | -0.52911                 |
|               | C4      | 0.13191                   | -0.13319                 |
|               | C1      | 1.8623                    | 0.24144                  |
|               | C2      | 0.093389                  | 0.14834                  |
|               | C3      | 1.8587                    | -0.23997                 |
|               | C4      | 0.088295                  | -0.14009                 |
| 2.0           | C1      | 1.9272                    | 1.2075                   |
|               | C2      | 1.2778                    | -1.4353                  |
|               | C3      | 2.0505                    | 0.4938                   |
|               | C4      | 0.38877                   | -1.4885                  |
| 2.5           | C1      | 1.8577                    | 0.53519                  |
|               | C2      | 0.68423                   | 2.4847                   |
| 3.0           | C3      | 1.9581                    | 0.29632                  |
|               | C4      | 0.52254                   | 0.99299                  |
6. Conclusions

The CFD Analysis of the flow-field indicates that the spacing ratio has dominant effect on the flow pattern and surface properties such as drag and lift coefficients. For spacing ratio of 1.5, the bottom shear layer of top row first cylinder impinges on downstream cylinder. The resulting flow interaction leads to uneven pressure distribution on surface of the cylinders. Hence lift forces generated on the cylinder C1 are significant. A similar behavior is observed on bottom row cylinders, but lift force acts in the opposite direction of that observed on top row cylinders. The lift coefficient of downstream cylinders is lower compared to those of upstream cylinders. For spacing ratio of 2.0 a similar behavior is observed. For spacing ratio of 2.5, top shear layer of top cylinder impinges on downstream cylinder. The resulting flow interaction amplifies the asymmetry in the pressure distribution in the near wake region. This results higher lift coefficient than that observed for spacing ratio of 1.5. Also the lift coefficient of downstream cylinder is comparable to its upstream cylinder. For spacing ratio of 3.0, the shear layers of top row upstream cylinder merge and impinge on downstream. The resulting flow interaction reduces the asymmetry in the pressure distribution on the upstream cylinder. This leads to higher lift coefficient for downstream cylinder than that of upstream. For all spacing ratios, drag coefficient of upstream cylinders is higher than their respective downstream cylinders. It is observed that the sensitivity of lift coefficient to spacing ratio is higher than that of drag coefficient.

Acknowledgments

The authors acknowledge CFD Laboratory, Department of Mechanical Engineering, SRMIST, Kattankulathur for providing computational resources for performing the numerical simulations.

Nomenclature:

| Symbol | Description |
|--------|-------------|
| CFD    | Computational Fluid dynamics |
| κ      | Turbulent Kinetic Energy, J/kg |
| ω      | Specific turbulent dissipation rate, 1/s |
| SST    | Shear Stress Transport |
| C_d    | Drag Coefficient |
| C_L    | Lift Coefficient |
| C_p    | Coefficient of pressure |

7. References

[1] William B. Hobbs and David L Hu 2012 Tree inspired piezoelectric energy harvesting Journal of Fluids and Structures 28 103-114
[2] Williamson and Govardhan 2004 Vortex-Induced Vibrations Ann.rev.Fluid Mech, 36 413-455
[3] Bearman 1984 Vortex shedding form oscillating bluff bodies Ann.rev.Fluid Mech 16 195- 222
[4] Sadatoshi Taneda 1956 Experimental Investigation of the wakes behind cylinders and plates at low Reynolds numbers Journal of the Physical Society of JAPAN 11 302-307
[5] M M Zdravkovich 1998 Review of interference-induced oscillations in flow past to parallel circular cylinders in various arrangements Journal of Wind Engineering and Industrial Aerodynamics 28 183-300
[6] Sumner, Richards and Akosile 2008 Strouhal number data for two staggered circular cylinders Journal of Wind Engineering and Industrial Aerodynamics 96 859-871.
[7] Feifei Tong, Liang Cheng, Ming Zhao, Tongming Zhou and Xiao-bo Chen 2014 The vortex shedding around four circular cylinders in an in-line square configuration *Physics of Fluids* **26** 024112

[8] A.T. Sayers 1990 Vortex shedding from groups of three and four equispaced cylinders situated in a cross flow *Journal of Wind Engineering and Industrial Aerodynamics* **34** 213-221.

[9] Thom A 1933 The flow past circular cylinder at low speeds *Proc. Royal Soc. A* **141** 651–66