Numerical investigation of flow field over four in-line circular cylinders for different spacing ratio

D S K Reddy\textsuperscript{1}, R Iyer\textsuperscript{1}, K Luthra\textsuperscript{1}, A Rana\textsuperscript{1} and P Kumar\textsuperscript{1}
\textsuperscript{1}Department of Mechanical Engineering, SRM Institute of Science and Technology, Kattankulathur, Pin code: 603203, TN, India.

E-mail: pankajkr@srmist.edu.in

Abstract. The objective of this paper is to study the effect of spacing ratio on the wake dynamics in an inline arrangement of four cylinders. Different spacing ratios, defined as ratio of distance between center of two cylinders and diameter of cylinder, are considered while Reynolds number (Re) kept constant. At the chosen Re, far wake is expected to transition to turbulence, and hence $\kappa$-$\omega$ 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. In the numerical solutions it is observed that, two counter rotating vortices are formed in the gap between the cylinders. These two vortices found to move towards downstream as the spacing ratio is increased. Also, the size of the vortices found increased. The two counter rotating vortices appears symmetric near cylinders, and hence lift coefficient observed to be very low for the first three cylinders. Lift coefficient observed on fourth cylinder found to one order of magnitude higher. The drag coefficient for all cylinders found to be positive except for the second cylinder which has negative drag for all spacing ratios.

Nomenclature:

\textbf{CFD} = Computational Fluid dynamics
\textbf{Cd} = Drag Coefficient
\textbf{CL} = Lift Coefficient
\textbf{Cp} = Coefficient of pressure
\textbf{SST} = Shear Stress Transport
$\kappa$ = Turbulent Kinetic Energy, J/kg
$\omega$ = Specific turbulent dissipation rate, 1/s

1. Introduction

Bluff bodies have been subjected to extensive research especially flow over circular cylinders due to many practical applications such as risers, off shore structures, bridge pillars which show the phenomenon of vortex shedding whose frequency might match the natural frequency of these structures resulting in potential failure of these structures. The shear layers which originate from flow separation point roll up in wake and finally sheds in cylinder downstream. The flow over circular cylinders have been subjected to lots of research over the last few decades. Mainly these studies have been done to find the relationship between the flow parameters such as velocity, pressure distribution, hydrodynamic forces & Strouhal number. Many of the researches have performed numerical studies on flow over single & multi cylinders in different arrangements. The important types being tandem and staggered arrangements to understand the effect of this geometry on various flow parameters. Also
there have been extensive researches performed to find out the effect of variation of non-dimensional parameter Reynolds number and the spacing ratio (S/D) between the cylinders on the hydrodynamic force coefficients i.e. drag and lift coefficients and also a non-dimensional parameter called Strouhal number used to quantify the vortex shedding occurring in the wake of the cylinder the frequency of which if matches the natural frequency of the structures then resonance might damage the structures which is why these studies are important and been performed.

Tong et.al. [1] investigated different cylinders in square arrangement at Re 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. Wake flow behind two tandem cylinders was studied numerically by Dekhordi et al. [2]. The variation of spacing ratio, drag and lift coefficients and Strouhal number are investigated. Reynolds number in the range between 100 to 200 are in laminar regime and higher values like $2.2 \times 10^4$ are consider as turbulent. If the spacing ratio is less than 3, then drag coefficient of second cylinder found to be negative, and vortices are formed downstream cylinder only. Sinhamahapatra et.al [3] studied the influence of spacing ratio in two tandem cylinders arrangement. Flow development is reported as a function of Reynolds number. For Re below 40, flow achieves steady state for any spacing ratio. When Reynolds number is 70, and spacing ratio is greater than 2, an unsteady flow is observed. For Reynolds number greater than 100, for all spacing ratio, flow found to be unsteady. At lower gaps the vortices formed by the front cylinder impinges over the rear cylinder which definitely influences its hydrodynamic forces and causing the drag and lift to oscillate at a higher frequency than upstream cylinder. Strouhal numbers are not much different for the two cylinders because of the uniform and in-sync vortex shedding from the cylinders.

Harichandan et. al [4] have investigated the simulation on multiple cylinders by creating unstructured mesh. The cylinders were placed in tandem as well as in-line. Reynolds number considered are 100 and 200, and spacing ratio chosen are 1.5 D and 3 D for side by side; 2 D and 5 D for tandem arrangement. It was found that the increased spacing reduces the interference effect between the two cylinders. For tandem arrangement, it is clear from the vorticity contours that vortex from upstream cylinder influences the hydrodynamic force coefficients of the downstream cylinders. When the spacing increases to 5 D, the upstream cylinder having no influence on downstream. Also, non-dimensional frequency was constant in any arrangement. 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. Further, 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 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 $0^\circ$ to $90^\circ$. They observed reattachment and gap flows were observed at smaller and higher spacing ratio. Vortex dynamics is much more complex for array of cylinders. The relationship between the forces and the phases of the vortex shedding has been studied numerically by Alama et al. [6].

The different spacing was investigated for Reynolds number 200 with tandem arrangement. They have used SIMPLE algorithm with second order upwind scheme in their numerical technique. The analysis was extended for dependence of the force coefficients on the phases of the shedding of the vortices. They observed lift coefficient fluctuation is higher in amplitude for cylinder 2 compared to rest cylinder. The value of drag coefficient is highest when they are in phase, minimum for antiphase. The effect of the phase lag is thus much less or negligible on lift coefficient of the downstream cylinder. Masami et al. [7] numerically investigated for the vortex formation (von Karman vortices) phenomenon in the wake of a cylinder and its effect on the oscillations caused on the cylinder. For the Reynolds number 50, the flow separated on the cylinder wall surface at about 130 degrees from clockwise direction. Increase in Reynolds number increases the length of vortex region and higher Reynolds number causes to form the vortices that last for a shorter duration which consequently
increased the distance between them. With the increase in Reynolds number over 1000 the flow in the wake becomes very irregular. The drag force was plotted with respect to time and it was observed that with low Reynolds number the fluid force just reached steady state quickly but as the Reynolds number became higher and the vortices begin to form the forces were observed to oscillate and became harmonic after a period of time. Sayers [8] has carried out experiments on cylinders with constant spacing cylinders at Reynolds number of $3 \times 10^4$ in range of spacing from L/D 1.5 to 5 and at inclination angle ($\alpha$) 0° to 180°. They observed that for four cylinders Strouhal number was higher than the single cylinder, due to array of cylinders behave as a single cylinder and at higher spacing wake of cylinder array is quite complex. Meneghini et al. [9] did numerical study on cylinder with different gap. They observed that up to the three times cylinder diameter downstream length, both cylinders behave as a single cylinder and perform shedding. After this gap, both cylinders shed independently. Ding et al. [10] studied the cylinders in tandem and side arrangement. They observed that up to gap ratio 2.5, flow is steady but with higher gap ratio flow becomes unsteady.

All the numerical studies have been carried out using different numerical schemes, many researchers have adopted different numerical models for solving the Navier-Stokes equation. Abaqus was used by Masami et al. [1] and a 3-D element called FC3D8 was used to model the fluid and a decoupled method was adopted to solve the Navier-Stokes Equation. RNG $k$-$\varepsilon$ was used Dekhordi et al. [2] to model turbulence and a staggered grid was used which was non-uniform in particular and the refinement was done in the zones closer to the cylinders. Harichandhan et al. [4] developed an unsteady viscous two-dimensional finite volume Navier stokes solver using a flux reconstruction technique. Alama et al. [6], have used SIMPLE algorithm with second order upwind scheme in their numerical technique. Their grid is structured mesh and they have used the commercial code ANSYS-FLUENT package. Ding et al. [10] have employed finite difference method to carry out their numerical work whereas Meneghini et al. [9] performed a numerical study using finite element method.

In the present numerical work, four cylinders are taken in series with different spacing ratio for Re 200 which is not explored in past. Pressure coefficient, drag and lift coefficient are measured around each cylinder. Flow physics are explained with help of velocity contour.

2. Simulation Methodology:

ANSYS – Fluent is used for simulating the flow field over the array of cylinders. Simulations are performed on unstructured grids, which are generated using ANSYS Workbench meshing tool. The incompressible RANS equation is solved with pressure-based algorithm under steady state condition. $k$-$\omega$ SST model is used to close the turbulence terms in RANS equation. This model is used to simulate the problem because of the satisfactory results it gives when it comes to capturing the viscous near-wall region and stream wise pressure gradients as was studied, suggested and reported by Rosetti et al. [13]. At inlet, uniform velocity is 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 grid in XY plane. Fine mesh is used at the near wall region of the cylinder wall. The grids are progressively stretched towards outlet boundary so that a coarse grid can be employed. The first grid 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 shows geometry of the array of cylinders, this arrangement has the four cylinders in the C1, C2, C3, C4 in tandem respectively in the same order. The free stream conditions are: Reynolds number is 200 velocity of 0.005 m/s. The cylinders have identical diameters and its value is 0.04 m. Computational domain size of 20 diameters in height and 40 diameters in width.
Figure 1. Schematic representation of four cylinders in tandem arrangement. Diameter of the cylinder is represented by 'D', and spacing between cylinders is indicated by 'S'.

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

Figure 3 shows reduction in the predicted drag coefficient (Cd) as a function of the iterations for a typical simulation on logarithmic scale. There is a rapid drop in Cd from 0 to 100 iterations. In the subsequent iterations, the reduction in the Cd is slower. The % change in Cd from 8000 to 10000 iterations is less than 0.1%. As the drag coefficient (Cd) 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
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 with Rajani et al. [11] and Thom et al. [12] figure 4 and the profile was found out to be in pretty good agreement with little variation that is likely due to the steady state simulation.

![Figure 4](image)

**Figure 4.** The Pressure distribution around the cylinder i.e. $C_p$ vs Theta: Rajani [11], Thom [12] & Present Work.

![Figure 5](image)

**Figure 5.** Effect of grid refinement on coefficient of pressure, (a) Cylinder 1, $C_1$, (b) Cylinder 2, $C_2$, (c) Cylinder 3, $C_3$, (d) Cylinder 4, $C_4$. 

4. Grid Convergence

To assess the grid point density to resolve flow gradients, a grid convergence study is performed for spacing ratio of 2. Simulations are performed by increasing the number of points by 50% from Gird 1 to Gird 2 and by 50% from Gird 2 to Gird 3. The near wall spacing is kept constant in all the three grids. In the circumferential direction, the cylinder is divided into 100 divisions, 160 divisions and 200 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 Figure 5. For cylinder $C_1$, there is not much change in the shape of the pressure distribution with the refinement in the grid which shows that the cylinder is facing the free stream so there is not much effect of grid refinement. For cylinders 2,3 and 4, the solution changes for entire 360 deg as the grid is refined. As these three cylinders faces the separated flow from upstream cylinders, the flow field is more susceptible to grid refinement. The solution obtained from coarse grid has varies a little bit from that of medium and finer grids. In case of medium and fine grids give a smoother variation of $C_p$. Again, the solution obtained from medium and fine grids are very closely identical. Over all, further refining the mesh from grid 2 to grid 3 does not yields any significant changes. Hence a grid 3 of 200 number of divisions is used for simulation.

4. Results

4.1 Variation of Velocity magnitude for different spacing ratio

Figure 6 shows computed flow field in terms of velocity contours as a function of spacing ratio. Left side figures show view of the entire computation domain whereas the right-side figures show enlarged view near the cylinders. The global flow features remain same in all the cases. These include deceleration of the freestream flow in the nose stagnation point, flow expansion in the downstream of the nose stagnation point. The expansion results in increase of velocity of as indicated by the red colour. The expansion envelopes the downstream three cylinders. The flow separation from cylinder $C_1$ results in the formation of wake region. A Similar flow separation occur from rest of 3 cylinders. Hence the gap between the cylinder is filled by wake flow. The shear layers from $C_4$ rolls up and result in the formation of vortex shedding. As the sparing ratio, increases the strength of vortex shedding decreases. Overall, the four cylinders are acting a single body.

Enlarged views near the cylinders reveal the effect of spacing ratio on local flow features. The shear layer originated from cylinder $C_1$, passes over cylinder $C_2$. The extent of impingement of shear layer onto $C_2$ is noticeable but not significant. A further enlarged view of the vector flow field around $C_2$ is shown in Fig.7. This clearly shows formation of two counter rotating vortices in the space between $C_1$ and $C_2$. These vortices deflect the shear layer upward so that impingement of upstream shear layer from $C_1$ onto downstream cylinder $C_2$ in minimal. The Cylinder $C_2$ will not face freestream, but it is encountered by the vortex flow from $C_1$. A part of the vortex flow deflects towards $C_2$, and moves along the surface of $C_2$. However, the flow direction is opposite to that of freestream. This bifurcated flow marches towards base stagnation point of $C_1$ and circulates around the counter rotating vortices. The gap flow between cylinder $C_2$ and $C_3$; $C_3$ and $C_4$ are similar.

Downstream of $C_4$, a complex behavior of flow can be observed. The two counter rotating vortices persist downstream of $C_4$ also. But the global shear layer layers from top and bottom side of the cylinders will converge farther downstream of $C_4$. This results in formation of another vortex. This vortex is placed in between the counter rotating vortices and merging point of shear layer. The center of this vortex lies close to the central axis of the cylinders.

As the spacing ratio increases, the two counter rotating vortices are prominent, and size of the vortices increases. The center of vortex also shifts towards downstream cylinder. The shape, size and orientation of single vortex that is formed downstream of $C_4$ also changes with respect to the spacing ration. For spacing ratio of 1.5, orientation of vortex in downward and it is regular circular shape, whereas for spacing ratio of 3.5, the orientation is towards upward and it is in the form of triangle.
4.2 Pressure coefficient (Cp) variations with spacing ratio

Coefficient of pressure (Cp) for $C_1$ 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. The flow separation region is marked by the low values of Cp, and it spans approximately from 60 deg to 270 deg. The flow separation angle is reducing with increase in the spacing between the cylinders this is essentially due to the fact that the shear layer impingement of the upstream cylinders is reducing with the increase in spacing so it’s affecting the separation of the boundary layers in the downstream cylinders. The effect of reduction in flow interaction or interference can also be seen in the graph as the increase in the range of Cp values with increased spacing. The pressure distribution curve shifts higher showing a reduced effect of interference.

![Figure 6](image)

**Figure 6.** Numerical visualization of the flow filed in terms of the velocity contours for Reynolds number 200: (a), (c), (e), (g) represent Velocity contours, and (b), (d), (f), (h) depicts magnified view near the cylinders. Vectors also shown to visualize the flow direction.
Figure 7. Velocity vectors of the upstream and downstream of cylinder C2. (a), (b), (c), (d) are the respective vector images for spacing ratios S/D= 1.5, 2, 2.5, 3 respectively.

Figure 8(b) shows the Cp variation on C2. The Cp values over 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. Also same is the cases for C3, C4 which are kept in the wake or low-pressure region of their respective upstream cylinders we can see the plot to be different from that of the C1. The trends observed for C3 and C4 are different though they also show an increase in values with increase in spacing. Apart from the first cylinder C1, the nose stagnation point is not occurring at theta=0 that’s because they are not facing the free stream but facing the low-pressure wake of the previous cylinders.
4.3 Comparison of Drag and Lift Coefficients

Comparison of drag and lift coefficients for different spacing ratio is presented in Table 1. For all the spacing ratios, drag coefficient for $C_2$ is found out to be negative which is in agreement with the literature works which says that the drag value will become positive above a critical value of $S/D=3$. This negative value is due to the $C_2$ being placed in the low pressure region of the $C_1$ cylinder the difference in the pressure between upstream and downstream is negative which ultimately results in negative drag which also means that this cylinder feels a force of attraction towards the $C_1$ unlike other cylinders which are being forced in the direction of the free stream. The drag experienced by $C_1$ in all the cases is higher than $C_2$, $C_3$, $C_4$ since it’s facing the free stream on its upstream side. The drag value for $C_3$ changes to positive and it further is greater for $C_4$ as can be seen in the reported values in the table. The magnitude of drag is continuously increasing with $S/D$ 2, 2.5, 3 which is obviously due to the reduction in the interference effects between the four cylinders.

![Cp variation with respect to angle for (a) cylinder C1 (b) cylinder C2 (c) cylinder C3 and (d) cylinder C4.](image)

The lift coefficients reported in the table depict that the lift generated is only considerable for the extreme downstream cylinder i.e. $C_4$. In fact, this observation is true for all the four spacing ratios. And the coefficients of lift reported for other three cylinders are much lower as compared to the $C_4$ which shows that the vortex shedding phenomenon is only occurring from the this cylinder since the other cylinder such as $C_1$, $C_2$, $C_3$ do not have sufficient space for the shedding of vortices by separation of the shear layers which is made pretty much clear in the vector images of these cylinder in figure 5 they are depicted by two contra rotating vortices in their downstream region.
Table 1. Computed Drag ($C_d$) and Lift ($C_L$) coefficients for cylinders at different spacing ratios.

| Spacing ratio | Cylinder | Coefficient of drag $C_d$ | Coefficient of lift $C_L$ |
|---------------|---------|---------------------------|--------------------------|
| 1.5           | C1      | 1.0910                    | -0.0022                  |
|               | C2      | -0.1982                   | 0.0019                   |
|               | C3      | 0.0521                    | -0.0062                  |
|               | C4      | 0.1438                    | -0.0260                  |
|               | C1      | 1.0680                    | 0.0008                   |
|               | C2      | -0.2027                   | -0.0048                  |
| 2             | C3      | 0.0904                    | -0.0034                  |
|               | C4      | 0.1768                    | -0.0186                  |
|               | C1      | 1.0497                    | 0.0017                   |
|               | C2      | -0.2115                   | 0.0017                   |
| 2.5           | C3      | 0.1336                    | 0.0022                   |
|               | C4      | 0.2108                    | 0.0194                   |
|               | C1      | 1.0327                    | 0.0022                   |
|               | C2      | -0.2041                   | 0.0049                   |
| 3             | C3      | 0.1664                    | 0.0053                   |
|               | C4      | 0.2240                    | 0.0117                   |

5. 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. The vortex dynamics and the hydrodynamic force coefficients were reported and studied and it was found out that the C2 experiences a negative drag but all other cylinders are having positive drag coefficient. The wake of the cylinders C1, C2, C3 are having stable contra rotating vortices which is clearly having an impact on the distribution of pressure over the cylinder surfaces which has also been plotted and analyzed with respect to the angle measured clockwise direction. The interference effects is more dominant for S/D=1.5 & 2 whereas the effect clearly has impact over the drag and lift forces and also the von Karman street is formed in the far wake of the extreme downstream cylinder in all the four spacing ratios, which sheds alternating vortices from its surface which is the reason for fluctuating forces.

Acknowledgments

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

6. References

[1] Tong F, Cheng L, Zhao M, Zhou T, & Chen X 2014. The vortex shedding around four circular cylinders in an in-line square configuration Phys. Fluids 26
[2] Dehkordi, B G, Moghaddam, H S & Jafari, H H 2011. Numerical simulation of flow over two circular cylinders in tandem arrangement J. Hydrondyn 23 114–126.
[3] Singha S and Sinhamahapatra K P 2010. High-resolution numerical simulation of low Reynolds number incompressible flow about two cylinders in tandem J. Fluids Eng. 132
[4] Harichandan A B & Roy A 2010. Numerical investigation of low Reynolds number flow past two and three circular cylinders using unstructured grid CFR scheme. Int. J. Heat Fluid Flow 31(2) 154–171.
[5] Zdravkovich M M 1988. Review of interference-induced oscillations in flow past to parallel circular cylinders in various arrangements J. Wind Eng. and Ind. Aerodyn. 28 183-300
[6] Md. Alam M, Zheng Q, Derakhshandeh J F, Rehman S, C Ji, Zafar F 2018. On forces and phase lags between vortex shedding from three tandem cylinders Int. J. Heat Fluid Flow 69 117-135.

[7] Sato M and Kobayashi T 2012. A fundamental study of the flow past a circular cylinder using abaqus/cfd, SIMULIA Design & Analysis Corporation

[8] Sayers A T 1990. Vortex shedding from groups of three and four equispaced cylinders situated in a cross-flow J. Wind Eng. and Ind. Aerodyn. 34 213-221

[9] Meneghini J R, Satara F 2001. Numerical simulation of flow interference between two circular cylinders in tandem and side-by-side arrangements. J. Fluids Struct. 15 327-350

[10] Ding H, Shu C, Yeo K S and Xu D 2007. Numerical simulation of flows around two circular cylinders by mesh-free least square-based finite difference methods. Int. J. Numer. Meth. Fluids 53 305-332.

[11] Rajani B N, Kandasamy A & Majumdar S 2009. Numerical simulation of laminar flow past a circular cylinder. App. Math. Model. 33 1228-1247

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

[13] Rosetti G F, Vaz G, and Fujarra A L C 2012. URANS Calculations for Smooth Circular Cylinder Flow in a Wide range of Reynolds Numbers: Solution Verification and Validation J. Fluid Eng. 134 1211-03