NUMERICAL SIMULATION ON FLOW PAST FOUR CIRCULAR CYLINDERS IN DIAMOND AND SQUARE ARRANGEMENT

THOTA SRINU¹, S. GIRISH², MONICA.T³ & B. DHANRAJ⁴

^{1,2,3} Assistant Professor, Department of Mechanical Engineering, MLR Institute of Technology,
Hyderabad, Telangana, India

⁴ Assistant Professor Department of Mechanical Engineering, Vardhaman College of Engineering, Shamshabad,
Hyderabad, Telangana, India

ABSTRACT

Flows past four cylinders, which are arranged in a diamond and square shape are simulated using ANSYS WORKBENCH. Flow is considered to be unsteady, incompressible and two dimensional and the examined Reynolds number is 5000. Simulations are performed for different gap to diameter ratio (L/D), ranging from 1.5 to 4, and for two incidence angles, $\alpha = 0^\circ$ and 45° . The flow characteristics, including the flow patterns, force parameters such as the drag and lift coefficients as well as wake oscillation frequencies (Strouhal numbers) were investigated. The force parameters are highly affected by the spacing ratio, three types of flow patterns were observed depending on the spacing ratio at two different angles of incidence. As the gap to diameter ratio varies from 1.5 to 3, more interference effects have been observed and further increasing the gap to diameter ratio, the flow pattern is similar to the flow past a single cylinder, by comparing both the arrangements square arrangement has been less interference effects.

KEYWORDS: Unsteady, Incompressible, Oscillation Frequencies & Interference Effects

Received: Nov 25, 2017; **Accepted:** Dec 15, 2017; **Published:** Jan 05, 2018; **Paper Id.:** IJMPERDFEB201834

INTRODUCTION

Circular cylinder has an important role in industry, because it is a common structure in many industries, it also has various engineering applications. Its practical applications in various engineering areas include heat exchanger tubes, suspension bridges, offshore platforms, boilers, cooling towers and automobile components. These structures are exposed to either water or air flow, so force induced forces are experienced by these structures, which could lead to failure of the structure. It is better to predict the flow behavior to avoid the damage of these structures. The study of flow past a single cylinder has been simulated by various researchers. Very less work was done in case of flow past an array of cylinders. It is very difficult to find the flow behavior in case of flow past an array of cylinders because of interference of vortices and wakes.

Numerical Modelling

Modelling has been a useful tool for engineering design and analysis. Modelling is the process of solving physical problems by appropriate simplification of reality. Most of the numerical models are based on the computational fluid dynamics (CFD), which contains numerical methodology to solve fluid flow problems and heat transfer problems in the form of software packages for simulation in the computers. CFD is an excellent and very powerful technique. It contains most of the applications in both industrial and non-industrial areas. Due to the

availability of high performance computers and graphical user friendly interface, it gained the interest among the engineers and researchers. CFD finds applications in hydrodynamics of submarine, ships, flow in rivers, channel, oceans and wind load over the buildings etc. CFD uses certain code and numerical algorithm to solve the problems which involves fluid flow. It is necessary to use the following three codes to get the desired solution. Those are

- Pre-processing
- Solver
- Post-processing

Pre-processing

In CFD it represents the input part for the flow problem. These are the few steps that the user must follow:

- Formation of geometry, representing the computational domain
- Meshing i.e. Splitting the domain in to small no of cells
- Applying suitable model required based on the physical phenomenon
- Giving the fluid properties
- Imposing boundary conditions

Solver

Fluent uses finite difference formulation technique, which is one of the three numerical methods in the CFD solver. The numerical methods are finite element method, finite difference method and finite volume method. Solver contains the following steps:

- Integration of fluid flow governing equations over the cells of the domain
- Discretization: conversion of integral equations into algebraic equations
- Interpolation of algebraic equations for getting the solution.

Post-processing

It represents the last part of the CFD programming tool. It contains recording of the results with versatile visualization technique, operator friendly interface and high graphics

This tool includes:

- Display of mesh and domain
- Visualization of contours
- Plotting vectors
- Streamline plotting
- Surface plots

Tang et al. [1] carried out investigations on flow past twin circular cylinders in tandem arrangement placed near a plane wall. A three step finite element method was used for solving two dimensional Navier-stokes equations. Simulations were carried out at a low Reynolds number of 200 for various dimensionless ratios of $0.25 < G/D < 2.0$ and $1 < L/D < 4.0$, where D is the cylinder diameter, L is the centre-to-centre distance between the two cylinders, and G is the gap between the lowest surface of the twin cylinders and the plane wall. Effect of G/D and L/D on the hydrodynamic force coefficients, Strouhal numbers and vortex shedding modes were observed. From the results three different vortex shedding modes of the near wake were observed. For various combinations of G/D and L/D , the hydrodynamic force coefficients and vortex shedding modes are quite different.

Kumar and Ray [2] carried out simulations by using the recently developed higher order compact scheme for $K=0.0, 0.1$ and for $Re=100, 200$, where Re is the Reynolds number and K is the shear rate based on cylinder side. r . Based on the results they observed that the vortex shedding phenomenon strongly depends on the Reynolds number as well as on the shear rate. In general average drag coefficient decreases with increase in the shear rate for fixed Re and for $K = 0.0 - 0.1$. Strouhal number decreases with increase in k . Investigations were carried out by **Lin Lu et al. [3]** for laminar flow past a circular cylinder having multiple control rods. Effects of Reynolds number, rod to cylinder spacing ratio, rod and cylinder diameter ratio and angle of attack were calculated. Based on lift and drag reduction four different flow regimes have been identified. As the spacing ratio increases at $Re=200$, the results for the case of six identical control rods show that the lift fluctuations on the main cylinder can be suppressed significantly for various diameter ratios, drag reduction on the main cylinder can also be achieved simultaneously.

Valipour et al. [4] simulated two dimensional, steady and laminar flow around and through a porous cylinder numerically, they have taken the range of Reynolds number and Darcy numbers as 1-45 and 10. Finite volume method is used for solving governing equations together with the boundary conditions, the effects of Reynolds numbers and Darcy numbers on pressure coefficient, wake structures and streamlines have been investigated. The results were compared with the solid and porous diamond square cylinders. Based on the results, they found that as the Darcy number increases the wake length and pressure coefficient decreases.

Han et al. [5] have done investigations on flow past four cylinders which are arranged as a square shape, Reynolds number was taken as 200. Spectral element method was used to carry out the simulations. Flow characteristics of two groups of cases were studied, one with an incidence angle of $\alpha = 0^\circ$ and other with $\alpha = 45^\circ$, here the parameter used is spacing ratio L/D which vary from 1.5 to 4. By changing the spacing ratios the flow characteristics including the flow patterns, statistical force parameters such as the drag and lift coefficients as well as wake oscillation frequencies (Strouhal numbers) have been investigated by Finite volume method. They kept the diameter of the downstream main cylinder as constant and varied the diameter of the upstream control cylinder. They have taken the input parameters as gap-to-diameter ratio (G/D) and the diameter ratio between the two cylinders (d/D). The gap between the control cylinder and the main cylinder (G) ranged from $0.1D$ to $4D$. They found that the gap-to-diameter ratio (G/D) and the diameter ratio between the two cylinders (d/D) have considerable effect on the drag and lift coefficients, pressure distributions around the cylinders, vortex shedding frequencies from the two cylinders and flow characteristics.

PROBLEM DESCRIPTION AND PROCEDURE

Computational Modelling

Computational fluid dynamics is a computer aided engineering tool. It can be designed for analysing the problem

involved in heat transfer, fluid motion etc. It gives good result and it has more advantages over experimental approach in terms of reduction of computational time and cost. The computer based software like Auto-Cad, Ansys and Solid works are used for optimisation of numerical models. The differential equations governing the flow are converted to transport equation for the numerical algorithm, which is later followed by the software for simulation purpose.

Numerical Analysis

Methodology: My present paper contains four cylinders which are arranged at 0° angle and 45° angle. Flows over these four cylinders are simulated using ANSYS WORKBENCH (FLUENT). Air is taken as a fluid. Flow is taken as two dimensional, incompressible and turbulent.

Problem Specification

Figure 1: Schematic Diagram of Computational Domain with Four Cylinders in Diamond Arrangement(0° Angle)

- Cylinder 3 is in downstream of cylinder 1 in tandem, while cylinders 2 and 4 are side by side arranged
- The spacing ratio L varies from $1.5D$ to $4D$ in the present simulation.
- The inlet boundary is at a distance of $8D$ from the centre point of the cylinder 1, while the outlet boundary is $24D$ away from the centre point of the cylinder 3; each lateral surface is $10D$ away from the centre point of the cylinders 2 and 4.
- Here the spacing ratio L/D varies from 1.5 to 4.

Figure 2: Schematic Diagram of Computational Domain with Four Cylinders in Square Arrangement 45° Angle

Table 1: Parameters are Considered

Re	Angle (Degrees)	L/D (Spacing Ratio)
500	0	1.5
		2
		2.5
		3
		3.5
		4
	45	1.5
		2
		2.5
		3
		3.5
		4

Governing Equations

The differential equation governing incompressible, viscous, unsteady and 2D-flow are the continuity and momentum equations. Continuity equation is the law of conservation of mass. It is expressed in Cartesian co-ordinates as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$$

u, v and w are the components of velocity along x, y and z directions. For incompressible flow density (ρ) is constant. So the above equation can be written as

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

For two dimensional flow

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

It is derived from the Newton's second law of motion. Forces that are acting on the particle of a fluid element are equal to the product of element mass and its acceleration along the flow direction. In the flow, momentum conservation is also called as Navier- Stokes equation.

For incompressible and two dimensional flow Navier-Stokes equation is written as follows:

x- Momentum equation:

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \frac{1}{\text{Re}} \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right]$$

y- Momentum equation:

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} + \frac{1}{\text{Re}} \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right]$$

Where, u and v are the velocity components

$$\text{Re} = \frac{\rho u D}{\mu}$$

μ is the dynamic viscosity of the fluid and ρ is the fluid density.

SIMULATION OF PRESENT WORK

For simulating my present work the following procedure has been followed: Click on the Ansys workbench icon. The workbench contains Toolbox. By double clicking on fluent option in toolbox the fluent window is opened in workbench as shown below.

Geometry Creation

Geometry creation is started by doubling clicking on the geometry which is said to be design modeller and select 2D in properties checkbox. After that by using sketching and modelling options, geometry is drawn in design modeller on XY plane. Here, geometry is flow domain of rectangular shape in which, four cylinders are arranged at 0° angle and 45° angle. Dimension option in the design modeller is used to give required dimensions to the geometry. We can also import geometry from other software like CATIA, CAD, solid works etc. In design modeller we can do both 2D sketching and 3D modelling. In the present work, 2D flow past a cylinder has been simulated.

Figure 3: Ansys Geometry Model for Diamond Arrangement (0° and 45° angle)

Mesh Generation

Mesh generation plays an important role in getting good results and good convergence. In this mesh generation process the computational domain is divided into a number of cells or small elements called grids. Generally we have two types of mesh generation: one is structured mesh and another one is unstructured mesh. With the structured grids, regular connectivity between the cells takes place and quadrilateral cell shape is the most common structure in the structured mesh. There is an irregular connectivity between the cells with the unstructured mesh, and triangular elements are most commonly used elements in this unstructured mesh, and it is one of the simplest types of mesh; it is always quick and easy to create. In the present problem, I have given the unstructured mesh for our solution domain. Discretization of the governing equation of flow is done at each cell or node to solve the Navier-Stokes equation. Therefore, it is important to give better mesh quality for obtaining good results and good convergence. With the mesh generation, better visualization of the boundary layer is

possible. Before generating mesh, we have to do grid independence test.

Grid Independence Test

Grid independence test is done to ensure that our solution doesn't depend on the grid size. The results of the problem can't change with the further increasing the grid size then the solution is said to be grid independent solution.

Grid independence test is done for my problem as shown below:

Figure 4: Variation of Drag Coefficient with no of Elements

After 42,266 elements, the solution didn't change even by increasing the number of elements. So, we have taken 42,266 elements for the present problem. The mesh quality is checked by using orthogonal quality of mesh. The range for the orthogonal quality is shown below:

Orthogonal Quality mesh metrics spectrum					
Unacceptable	Bad	Acceptable	Good	Very good	Excellent
0-0.001	0.001-0.14	0.15-0.20	0.20-0.69	0.70-0.95	0.95-1.00

Mesh Quality:

Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.

Minimum Orthogonal Quality = 5.08692e-01

Maximum Aspect Ratio = 6.91112e+00

Once a good quality mesh is achieved, we will go for solver for further analysis. If good quality mesh is not achieved again, we need to refine, re-size or change the type of mesh for e.g. Triangular, quadrilateral or mixing of both elements to achieve good quality. Finite volume method, Finite difference method and Finite element methods are used for discretising the domain in to mesh. In the present work, Finite volume method is employed to generate unstructured mesh for the computational domain.

Named Selections

We should give names to our flow domain by using create named selection option in this mesh window. Names are given to my flow domain as shown below, these named selection are used in setup to give boundary conditions. So, it is very important to give named selections to our flow domain.

Figure 5: Names Given to the Computational Domain

Results come under post-processing type. From results, we obtain contours of velocity, Pressure, Stream lines etc. Better visualization of flow field is possible with these contours. Animation of contours and streamlines are also generated from this option.

For diamond arrangement (0° angle) solution is converged as shown below:

Figure 6: Convergence of Solution Along with Scaled Residuals for Diamond Arrangement (0° Angle)

For square arrangement (45° angle) solution is converged as shown below:

Figure 7: Convergence of a Solution along with Scaled Residuals for Square Arrangement (45° Angle)

RESULTS AND DISCUSSIONS

Flow over a circular cylinder is simulated for different spacing ratios and for two different angles of arrangement those are 0° and 45° . Variation of coefficient of drag with different spacing ratios, variation of lift coefficient with different spacing ratios, variation of Strouhal number for all cylinders with different spacing ratios and for two different incidence angles were plotted in this chapter. Velocity contours, pressure contours and stream lines are also presented at each spacing

ratio for two different angles of arrangement.

Validation for Single Cylinder (Laminar Flow)

Figure 8: Computational Domain for Single Cylinder

Figure 9: Ansys Geometry Model Figure 4.3 Mesh given to the Computational for Single Cylinder Domain

The results for the single cylinder are shown below:

Figure 10: Convergence of a Solution for Single Cylinder

Contours and Streamlines:

Figure 11: Velocity Contours

Figure 12: Streamlines

Comparison of Results with other Literature Works

Results of the present work are same as the other literature work results. Validation of software is done

Table 2: Comparison of Present Results with Other Literature Works for Single Cylinder (Laminar Flow)

Case	C_D (Drag Coefficient)	St (Strouhal Number)
Present work	1.39	0.167
Meneghini et al.[10]	1.37	0.165
Kang[15]	1.33	0.165

Drag Coefficient for Diamond Arrangement (0° Angle)

Table 3: Values of Drag Coefficient for Diamond Arrangement (0° Angle) at Different Spacing Ratios

L/D	C_{D1}	C_{D2} & C_{D4}	C_{D3}	Single Cylinder
1.5	0.618	0.838	0.246	0.98
2	0.489	0.842	0.370	0.98
2.5	0.528	0.940	0.143	0.98
3	0.630	1.13	0.320	0.98
3.5	0.680	1.24	0.089	0.98
4	0.738	1.14	0.099	0.98

Figure 13: Variation of Drag Coefficient (C_D) with Spacing Ratio (L/D) for Diamond Arrangement (0° Angle)

Figure 14: Variation of Lift Coefficient with Spacing Ratio for Square Arrangement (45° Angle) and for Cylinders 1 & 2

CONCLUSIONS

- Three types of flow patterns were observed, depending up on the spacing ratio.
- At small spacing ratio ($L/D=1.5$), the overall flow past a four circular cylinders is similar to the flow past a single cylinder, because gap flow is completely suppressed at small spacing ratio and downstream cylinders are completely submerged in the wakes which were formed behind the upstream cylinders
- At the spacing ratio of 2.5 to 3.5, transition of flow pattern takes place from narrow gap flow to vortex impingement flow, high interactions and very complicated flow was developed.
- With the spacing ratio from 3.5 to 4, vortex impingement flow was developed, means vortices were fully developed and shed down from the cylinders
- The drag coefficient and Strouhal number are high in diamond arrangement, as compared to square arrangement because of less interference effects in square arrangement

Scope for Future Work

- Work can also be done for flow past four circular cylinders arranged at an incidence angles of 30° and 60°
- Simulations are also carried out by varying cross sections (square, triangle and polygons)

REFERENCES

1. Guo-qiang Tang, Chuan-qi Chen, Ming Zhao and Lin Lu, "Numerical simulation of flow past twin near-wall circular cylinders in tandem arrangement at low Reynolds number", *Journal of Water Science and Engineering*, 2015, Vol.8(4), pp.315-325.

2. Atendra Kumar and Rajendra Ray, "Numerical Study of Shear Flow Past a Square Cylinder at Reynolds Numbers 100, 200", *International Conference on Computational Heat and Mass Transfer*, 2015, Vol.127, pp.102 – 109.
3. Lin lu, Ming-ming Liu, Bin Teng, Zhen-dong Cui, Guo-qiang Tang, Min Zhao and Liang Cheng, "Numerical investigation of fluid flow past circular cylinder with multiple control rods at low Reynolds number", *Journal of fluids and structures*, 2014, Vol.48, pp.235-259.
4. Mohammad Sadegh Valipour, Saman Rashidi, Masoud Bovanda and Reza Masoodi, "Numerical modeling of flow around and through a porous cylinder with diamond cross section", *European Journal of Mechanics B/Fluids*, 2014, Vol. 46, pp.74–81.
5. Zhaolong Han, Dai Zhou, Xiaolan Gui and Jiahuang Tu, "Numerical study of flow past four square-arranged cylinders using spectral element method", *Journal of computers and fluids*, 2013, Vol.84, pp.100-112.
6. Yong-tao WANG, Zhong-min YAN and Hui-min WANG, "Numerical simulation of low-Reynolds number flows past two tandem cylinders of different diameters", *Journal of water science and engineering*, 2013, Vol. 6(4), pp.433-445.
7. Chilakala venkatareddy, Munaganuri, Anil Kumar & Junaid Farooq, *Hysteresis Damping in a Steam Turbine Blade by Using ANSYS and Hypermesh*, *International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)*, Volume 3, Issue 1, January - March 2013, pp. 181-188
8. Yan Bao, Qier Wu and Dai Zhou, "Numerical investigation of flow around an inline square cylinder array with different spacing ratios", *Journal of computers and fluids*, 2012, Vol.55, pp.118-13.
9. A.B Harichandan and A. Roy, "Numerical investigation of flow past single and tandem cylindrical bodies in the vicinity of a plane wall", *International Journal of Heat and Fluid Flow*, 2012, Vol.33, pp.19-43.
10. Dehkordi Behzad Ghadiri, Moghaddam Hesam Sarvghad and Jafari Hamed Hourri, "Numerical simulation of flow over two circular Cylinders in tandem arrangement", *Journal of hydrodynamics*, 2011, Vol.23, pp.114-126.
11. Zou Lin, Lin Yu feng and Lu Hong, "Flow patterns and force characteristics of laminar flow past four cylinders in diamond arrangement", *Journal of hydro dynamics*, 2011, Vol.23(1), pp.55-64.
12. S. Berrone, V. Garbero and M. Marro, "Numerical simulation of low-Reynolds number flows past rectangular cylinders based on adaptive finite element and finite volume methods", *Journal of Computers & Fluids*, 2011, Vol.40, pp.92–112.
13. Atal Bihari Harichandan and Arnab Roy, "Numerical investigation of low Reynolds number flow past two and three circular cylinders using unstructured grid CFR scheme", *International Journal of Heat and Fluid Flow*, 2010, Vol.31, pp.154-171
14. Manish Mehta & P M George, *Elastodynamic Analysis of Four Bar Mechanism Using Matlab and Ansys WB*, *International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)*, Volume 2, Issue 3, August - September 2012, pp. 76-82
15. Yan Bao, Dai Zhou and Cheng Huang, "Numerical simulation of flow over three circular cylinders in equilateral arrangements at low Reynolds number by a second-order characteristic-based split finite element method", *Journal of computers and fluids*, 2010, Vol.39, pp.882-899
16. B.N. Rajani, A. Kandasamy and Sekhar Majumdar, "Numerical simulation of laminar flow past a circular cylinder", *Applied Mathematical Modelling*, 2009, Vol.33, pp.1228-1247.
17. J.R. Meneghini, F. Saltara, C.L.R. Siqueira and J.A. Ferrari Jr., "Numerical simulation of flow interference between two circular cylinders in tandem and side-by-side arrangements", *Journal of fluids and structures*, 2001, Vol.15, pp.327-350.
18. Kang, "Characteristics of flow over two circular cylinders in a side-by side arrangement at low Reynolds number", *Physics of Fluids*, 2003, Vol.17(4), pp. 561-577.