CFD Hydrodynamic Analysis of Fluid Flow past Three Cylinders in Triangular Arrangement

Abhishek Shukla[1] Dr. Anil Kumar Tiwari[2]
[1,2]Department of Mechanical Engineering
[1]SSTC (SSEC) BHILAI (CG) [2]NIT, RAIPUR (C.G)

Abstract— This article provides an overview on the hydrodynamic analysis of flow over three cylinders in triangular arrangement by applying the Finite Volume approach. The paper includes the flow parameters like varying orientation of cylinders, the distance between the cylinders to study the effects like dynamic Pressure variation in flow separation, wall fluxes, streamlines with varying governing parameters. The two different orientations for 0º and 90º are compared with common flow velocity. The fluid used is benzene (C_6H_6).The results obtained are summarized comparing the different positioning. The comparison with their agreements and differences are clearly explained.

Key words: Orientation, CFD, Turbulence, Dynamic Pressure

I. INTRODUCTION

Flow over cylinders has been a matter of interest from a long period of time for various engineering applications as Basins of oil extractions, Flow in channels, Flow through pipes, Flow associated with submarines, aerodynamics etc. It is the study focused to simulate and analyze the situation arising for complex flow conditions For various engineering calculation and analysis it is necessary to have an exact overview of the effect of various parameters and their associated effect on the flow dependent conditions. The present study involves numerical simulation of flow across three circular cylinders in triangular arrangement. The simulation for flow across this arrangement finds investigative opportunities to study and analyze Dynamic Pressure variation, effect of Turbulence, Arrangement of cylinders, Orientation of Cylinder

II. LITERATURE REVIEW

A Numerical simulation of laminar flow past a circular cylinder" by B N Rajani, A.Kandasamy and Sekhar Majumdar,(1997)Department of Mathematical and Computational Science NIT Karnataka, Bangalore focuses on analysis of two dimensional flow past circular cylinder in laminar flow regimes. It involves finite volume discretization method. Computational results were validated against measurements for mean surface pressure, skin friction coefficients, size and strength of recirculating wake. It also deals with Strouhal frequency of vortex shedding. This paper helps to understand the basic flow past a cylinder , flow separation and vortex shedding process.

Laminar flow past an oscillating circular cylinder in cross flow by Chang Yeol Lee, Jeong Hu Kim, Ryong Park,(2001). This paper provides the numerical study of two dimensional laminar flow past circular cylinder forced to oscillate transverse to the free stream direction.

Cylinder oscillation frequency was ranged between 0.8 to 1.2 of natural vortex shedding frequency. The oscillation amplitude was extended to 50% of cylinder diameter at Re=185 showing typical 2 dimensional vortex shedding .Quantitative analysis about flow variables and distribution of wall vorticity on cylinder surface, drag and lift coefficients was highlighted .This paper helps to understand the effects of vibration induced flow.

The mixing of a viscous fluid in a layer between cylinders by A.G. Petrov(2008)in Journal of Applied Mathematics and Mechanics .This paper deals with motion of an incompressible viscous fluid in a thin layer between two circular cylinders, inserted into one another, with parallel axes is investigated. The cylinders rotate relative to one another about an axis parallel to the axes of the cylinders. This paper provides information regarding fluid particle interaction between Cylinders results were plotted and shown for Re=100,328 and 510 showing the Pressure fields, effect of viscosity on wake formation , Drag and lift.

Modelling of water wave interaction with multiple cylinders of arbitrary shape by Longbin Tao, Subrata, Chakrabarti (2009).This paper describes the development of an efficient numerical model for linear waves interaction with cylindrical structures of arbitrary shapes. The two-dimensional Flow equation is firstly applied in the circumferential direction, so that the governing partial differential equations are transformed to an ordinary matrix. The wave forces and run-ups are presented for a single and multiple cylindrical structures of different cross sectional shapes. Influences of the incident wave parameters and structural configurations on the hydrodynamics were examined

Various other researchers like Rajeev K. Jaiman , Subhankar Sen, Pardha S. Gurugubelli on "Freely vibrating square cylinder" , C.W. Li, M.L. Zhang on "Numerical modeling of shallow water flow around arrays of cylinders", E.E.M. Olsson , L.M. Ahrne´ A.C. Tragardh on "Flow and heat transfer from multiple slot air jets impinging on circular cylinders" and many other researchers on different topics have provided their valuable view on the flow associated with cylinders.

III. MATHEMATICAL MODELLING

For the flow involving flow past cylinders the Navier–Stokes equations can be transformed for applying the finite volume method:

$$\rho \frac{Du}{Dt} = -\frac{\partial p}{\partial x} + div(\mu gradu) + S_{Mx} \quad (1)$$

$$\rho \frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + div(\mu gradv) + S_{My} \quad (2)$$

$$\rho \frac{Dw}{Dt} = -\frac{\partial p}{\partial z} + div(\mu gradw) + S_{Mz} \quad (3)$$

All rights reserved by www.ijsrd.com

The basic governing equations involve the Continuity, X and Y momentum and the Energy equation which are summarized as below:

Continuity Equation

$$\frac{\partial \rho}{\partial x} + div(\rho u) = 0$$

X-momentum Equation

$$\frac{\partial(\rho u)}{\partial x} + div(\rho uu) = -\frac{\partial p}{\partial x} + div(\mu gradu) + S_{Mx} \quad (4)$$

Y-momentum Equation

$$\frac{\partial(\rho v)}{\partial y} + div(\rho vu) = -\frac{\partial p}{\partial y} + div(\mu gradv) + S_{My} \quad (5)$$

Energy Equation

$$\frac{\partial(\rho i)}{\partial t} + div(\rho iu) = -pdivu + div(kgradT) + \Phi + S_i \quad (6)$$

Turbulent kinetic energy:

$$(\rho k)\partial t + \partial(\rho kuj)\partial xj = \partial\partial xj[(\mu + \mu t\sigma k)\partial k\partial xj] + \eta t\partial ui\partial xj(\partial ui \partial xj + \partial uj\partial xi) - \rho\varepsilon \quad (7)$$

Turbulent dissipation energy:

$$(\rho\varepsilon)\partial t + \partial(\rho\varepsilon uj)\partial xj = \partial\partial xj[(\mu + \mu t\sigma\varepsilon)\partial\varepsilon\partial xj] + C1*\varepsilon k\eta t\partial ui\partial xj (\partial ui\partial xj + \partial uj\partial xi) - C2\rho\varepsilon 2k \quad (8)$$

Where

$\mu t = \rho C\mu k2\varepsilon$

$C1* = C1 - (1 - \eta/\eta 0)1 + \beta\eta 3$

$\eta = (2E\imath j * E\imath j)12k\varepsilon$, $E\imath j = 12[\partial ui\partial xj + \partial uj\partial xi]$ $C\mu = 0.085$,

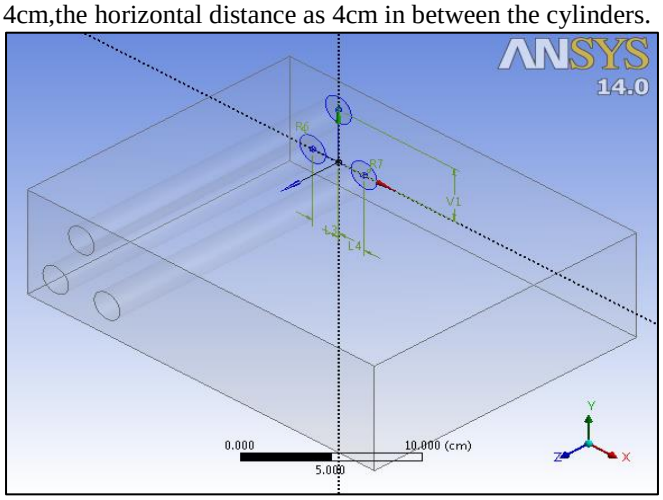
$C1 = 1.42$, $C2 = 1.68$, $\beta = 0.012$, $\eta 0 = 4.38$

For incompressible fluid the density remains constant.

The mathematical modeling includes the arrangement for three cylinders and with 0^0 Orientation and 90^0 orientation of three taking pipe diameter 1cm,Velocity of flow (2m/s), the altitude distance (4cm) in an triangular arrangement.

IV. METHODOLOGY

The ANSYS 14.0 finite volume element program was used for analyzing flow across the triple cylinder arrangement. For the purpose to be achieved, the geometry was created, with cylinder diameters of 1 cm,the altitude distance as 4cm,the horizontal distance as 4cm in between the cylinders.



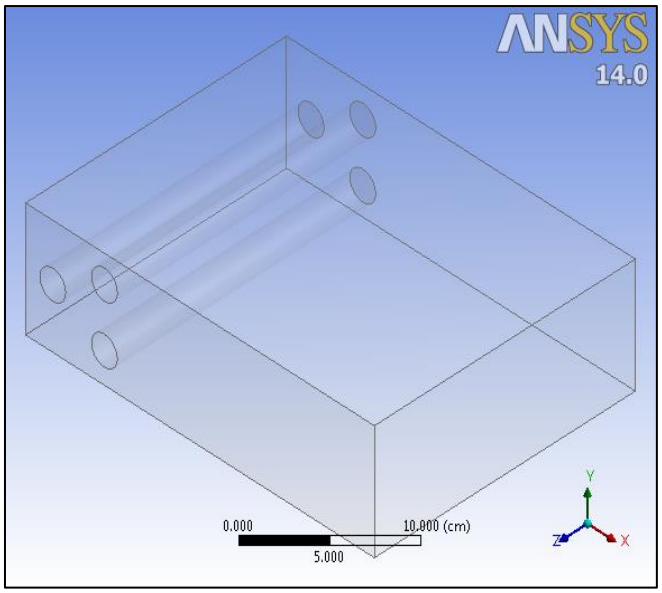


Fig. 1: Model Geometry

Tetrahedral Meshing with coarse and fine at solid-fluid boundary was taken to clearly understand the flow with desired accuracy and precision A 20-node three-dimensional structural solid element was selected to model the $0^0$and 90^0 orientation arrangement .The cylinders were discretized into 6624 elements with 5768 nodes. For both the arrangements The .Inlet Pressure was taken 300Kpa ,initial velocity as 2m/s , Exit Pressure as 100 Kpa, The fluid density is 875 Kg/m^3 and viscosity as 0.00589 N-s/m^2 .

V. RESULT AND DISCUSSION

The governing equations of the problem were solved, using a finite Volume method (FVM). The hydrodynamic characteristics were calculated. The results obtained are presented and discussed in form of contours.

The contour plots are clear to understand and depict a clear picture of the varying parameters through out the flow.

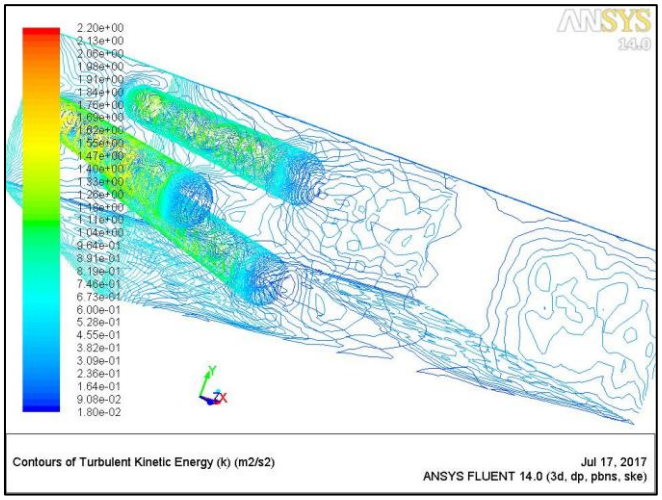


Fig. 2: contour plot of turbulent kinetic energy of 0^0 Orientation.

All rights reserved by *www.ijsrd.com*

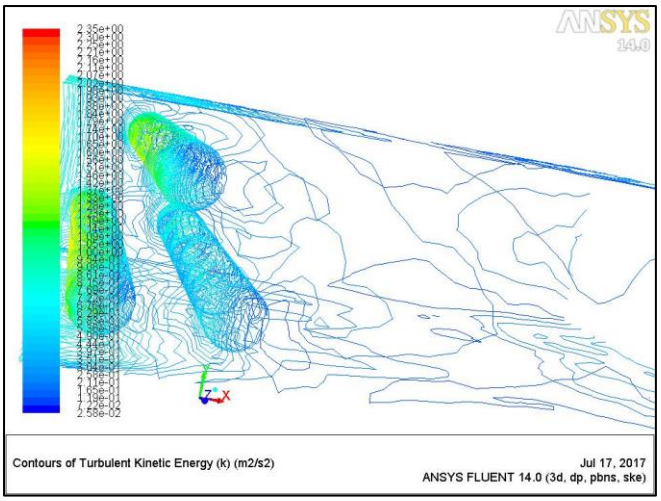


Fig. 3: contour plot of turbulent kinetic energy of 90^0 Orientation

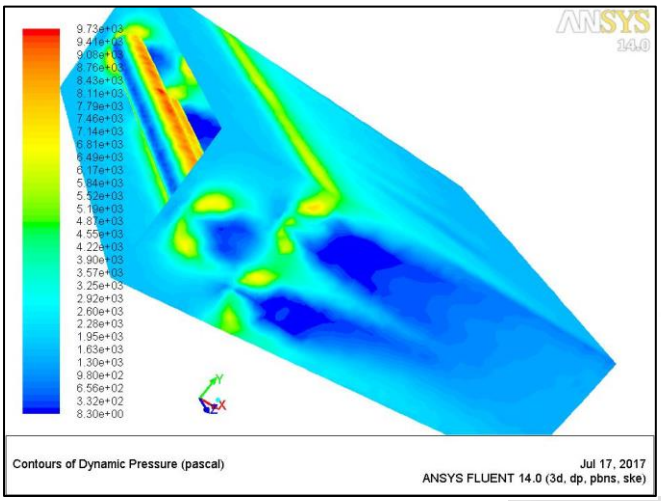


Fig. 4: contour plot of Dynamic pressure of 0^0 Orientation

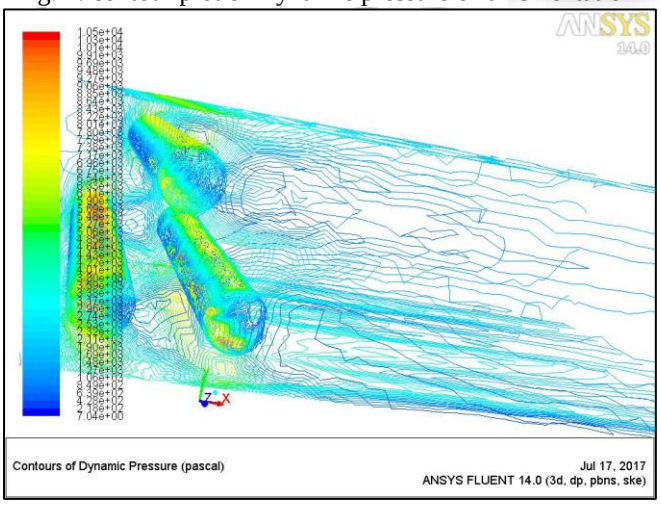


Fig. 5: contour plot of Dynamic pressure of 90^0 Orientation

Figure 2 represents the turbulent kinetic energy in a 0^0 Orientation. It is seen that turbulent kinetic energy reaches maximum value when fluid strikes the first cylinder across the flow. The turbulence kinetic energy then is nearly equal for the top cylinder and the bottom cylinder. This is due to the fact that separation occurs more for the first impact as compared to the following ones.

Figure 3 depicts the turbulent kinetic energy in a 90^0 Orientation. It shows that second bottom cylinder has nearly negligible turbulence kinetic energy as compared to the first cylinder and the top cylinder. The fluid nearly flows smoothly over the second bottom cylinder with very less separation.

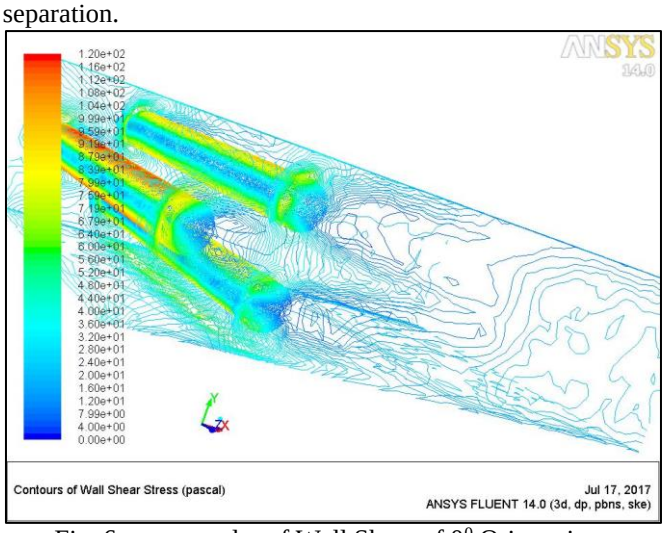


Fig. 6: contour plot of Wall Shear of 0^0 Orientation

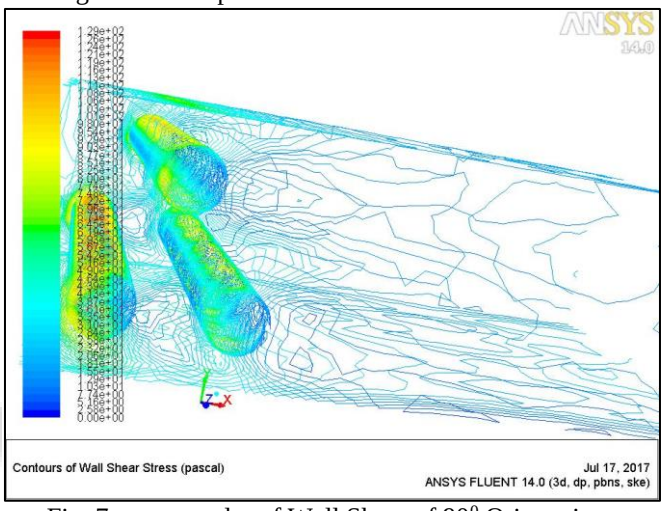


Fig. 7: contour plot of Wall Shear of 90^0 Orientation

Figure 4 shows contour of Dynamic Pressure in 0^0 Orientation. The dynamic pressure rises abruptly at top of the first cylinder when fluid strikes. The bottom portion however shows relatively low value, The fluid then passes on to the next two cylinder maintaining a nearly constant value for both the cylinders

Figure 5 represents contour of Dynamic pressure in 90^0 Orientation .It shows that in this arrangement also the first cylinder has highest value of dynamic pressure at the top, but here the second and third cylinder also contribute higher values of dynamic pressure at their top and bottom as compared to the sides across flow.

Figure 6 shows the wall shear stress contour for the 0^0 Orientation. It can be clearly seen that wall shear stress reaches maximum values for both top and bottom of the first cylinder. The second cylinder at the top portion also experiences high values of shear stress. The bottom cylinder however has very little values .

Figure 7 represents the wall shear stress for 90^0 Orientation. This arrangement experiences highest value of shear stress for the first cylinder but here, the top and bottom cylinders following it also experience considerable values of shear stress. This behavior is quite different from the other two parameters taken into analysis.

All rights reserved by www.ijsrd.com

VI. CONCLUSION

The turbulence kinetic energy for both the arrangements shows that fluid separation fluid is more for the 0^0 orientation as compared to the 90^0 orientation. It is due to the fact that fluid particles impacting on the first cylinder in 0^0 orientation experience more directional change while in 90^0 orientation they touch the first cylinder but not with much impact.

On comparing the performance in terms of dynamic pressure it is found that $.0^0$ orientation provides check against rising dynamic pressure for the first striking cylinder where as in 90^0 arrangement the second and third cylinder also experience higher values of dynamic pressure. This is due the change in angle of altitude and horizontal distance.

The analysis of wall shear stress for the two arrangements has varying results as in 0^0orientation although the first cylinder and the second top cylinder experience higher values but the bottom cylinder gets very low values. This is due to fluid passing smoothly over it as its particle path is not much disturbed. In 90^0orientation all the cylinders get proper fluid exposure and so the shear stress is quite evenly for them. The fluid path reflects that more is flow impact, more will be shear stress values.

Both the arrangements find their applications in various flow required conditions. The study of the physics of the flow for both arrangements helps to decide their viability and feasibility in different areas.

REFERENCES

[1] B N Rajani, A.Kandasamy and Sekhar Majumdar,(1997) "A Numerical simulation of laminar flow past a circular cylinder" Applied Mathematical Modelling ,33(2002),1228-1247
[2] Chang Yeol Lee, Jeong Hu Kim, Ryong Park(2001). " Laminar flow past an oscillating circular cylinder in cross flow",Journal of marine science and Technology ,Vol 18 No-3,361-368
[3] Vinod Kumar , Sanjay Mittal, 2004,The Seventh asian Congress of Fluid Mechanics,Dec 8-12, Chennai
[4] A.G. Petrov,2008," The mixing of a viscous fluid in a layer between rotating eccentric cylinders",Journal of Applied Mathematics and Mechanics",Vol 72,536-549
[5] Atul Sharma ,Vinish Gada2011," On A Novel Dual – Grid Level-Set Method For Two Phase Flow Simulation ",Taylor and Francis Publication
[6] Rajeev K. Jaiman , Subhankar Sen, Pardha S. Gurugubelli,2014,
[7] "A fully implicit combined field scheme for freely vibrating square", Computers & Fluids 112 (2015) 1–18 Vol 88, 563-571
[8] E.E.M. Olsson , L.M. Ahrne A.C. Tragardh 2004 "Flow and heat transfer from multiple slot air jets impinging on circular cylinders",Journal of Computational physics ,Vol 201(2008),
[9] J.D Anderson,2005"Computatinal Fluid dynamics",Department of Aerospace Engineering ,University of Maryland
[10] Ferzigar and Peric,2005,"Computatinal Methods for Fluid Dynamics",Stanford University
[11] Hoffman and Chiang,2000,Computatinal Fluid Dynamics, Engineering Educational System Publication
[12] Chang jun Zhu,2009,Application of Fluid Mechanics,McGraw Hills publication

All rights reserved by www.ijsrd.com