Numerical analysis of flow structures behind the bluff body at different aspect ratio

To cite this article: Aditya Sharma et al 2018 IOP Conf. Ser.: Mater. Sci. Eng. 402 012056

View the article online for updates and enhancements.

You may also like

- Comparison of modeling methods and parametric study for a piezoelectric wind energy harvester
  Liya Zhao, Lihua Tang and Yaowen Yang

- Classification of gas–liquid flow patterns by the norm entropy of wavelet decomposed pressure fluctuations across a bluff body
  Zhiqiang Sun, Yanping Chen and Hui Gong

Piezoelectric wind energy harvesting subjected to the conjunction of vortex-induced vibration and galloping: comprehensive parametric study and optimization
  Kai Yang, Kewei Su, Junlei Wang et al.
Numerical analysis of flow structures behind the bluff body at different aspect ratio

Aditya Sharma¹, Pankaj Kumar¹ and Santosh Kumar Singh*¹

¹ Department of Mechanical Engineering, SRM Institute of Science and Technology, Kattankulathur, Tamil Nadu-603203, India
* Corresponding author: santoshkumarsingh.k@ktr.srmuniv.ac.in

Abstract. The present paper aimed for the numerical study on the flow structures behind the bluff body with different shapes. The variation of shapes like circular to elliptical was obtained by changing the major to minor axis ratio while keeping the Reynolds number constant. This numerical study was done with help of finite volume method using commercial software Ansys-Fluent at the lower subcritical Reynolds number, i.e., 5000 and 10000 for comparison of different parameter variations separately. As the earlier studies reveals the significant effect of blockage ratio on flow features on circular cylinder, this work will be extended for different aspect ratio. The basic results, which are validated against the published results, shows the flow feature are highly sensitive to shape and blockage ratio even for different shapes. The mean pressure and turbulent kinetic energy for different blockage and aspect ratio is presented and found the strong interactions between them.

Keywords. Bluff body, cylinder, Drag coefficient, subcritical Reynolds number.

1. Introduction

Over the last few decades, the knowledge of fluid flow over bluff bodies has been substantially ascertained for various aspects, mainly owing to the engineering significance of structural design, heat transfer, and acoustic emissions. However, flow around a circular cylinder still remains a challenging problem due to complex unsteady dynamics of the cylinder wake flow. Recent advances in numerical and experimental techniques have facilitated the investigation of such complex flow structures around bluff bodies. A cylindrical bluff body is prioritized as geometry of concern for its similarity to most engineered structures and having most focused research developments. Aero dynamicity of bluff bodies plays a crucial role in determining the disturbed flow characteristics. Research by Gera [1] flow past over a square cylinder for wake behaviour determining Reynolds number range for vortex shedding at zero incidence angle. Prasenjit and Das [2] found, for flow past a rounded edge square cylinder, the importance of shape over boundary layer thickness and displacement thickness. The conclusion that proportionality between radius and thickness states it as a character of circular cylinder due to the strong pressure gradient and weak wall shear stress developed around it. A closed type of conic section contributes towards ease of sincere certainty in interpolation of results. Therefore circular cross section cylinder adapts for the study of flow structure behind the bluff body with different shapes. The shape is
also in consensus with the common feature of disturbed flow around all bluff bodies, i.e. development of similar flow structures in the separated region.

Flow structures over the bluff bodies is extensively affected by Reynolds number [3]. The dimensionless quantity, quantifies the relative contribution and interaction of inertial forces to viscous forces of liquid for a particular flow condition, predicting onset of turbulent flow. Especially for a circular cross section cylinder, complexity of transition intensifies with increase in Reynolds number. Convolution of subcritical flow region is due to simultaneous prevailing of laminar and turbulent regimes. The modelling become more complicated due to the presence of intermittent turbulence character in wake flow have been mentioned in detail by Doolan [4]. For its immense vitality in engineering application separated flow structure are widely studied and remains epitome of significant research. Flow separation characteristics are paramount in respect to Reynolds number for analysis of flow transitions, which leads to awareness of vortex shedding character. For expertise in flow regime confined in gradually increasing Reynolds number sets, disturbance-free flow is thoroughly studied by minimizing influencing parameter for actual flow, accounted by Morkovin [5]. With increasing Reynolds number laminar flow separates at adverse pressure gradient point (separation point) in order to reattach at point of confluence. With increase in Reynolds number (Re) further, prompts acute vorticity and eddies formation and shedding which makes turbulent transition in whole of the flow, initiating at reattached region and followed in succession by wake region, free shear layer, and finally in boundary layer. This transition and vortex shedding in subcritical and supercritical region are separately studied in research presented by Doolan [4] and Roshko [6]. Results of experimental investigation on a circular rotating cylinder by Labraga et al. [7] mainly concern the location of separation point, shows shift in vortex shedding frequency as dimensionless rotation rate increases. A deliberate concern over the study of the development of the laminar separation bubble, mentioned in above latter paper, is explained by Miozzi et al. [8].

Wall blockage is an important paramter that affects transition in both boundary and free shear layer. The confining surrounding restricts the flow sidewise and impose interfering pressure gradient. For small gap to diameter ratio blockage becomes a governing parameter. Aspect ratio or the ratio between different axes of cross section has effects on drag coefficient and base pressure coefficient comparable to that of blockage. It becomes a governing parameter for a short finite cylinder with free end. Zdravkovich [3] presents a detailed review on wake transition and basics for relevant influencing parameters effect. Major and minor axis contribution as governing parameters are also analysed through research contributed by Jelita et al. [9] on heat transfer over temperature gradient of elliptical cylinder showing symmetric distribution heat transfer to right angle of major axis. Investigation on disturbed flow around bluff body equivalent to that of fluid flow past a stationary body, is also presented through study of body motion in a fluid at rest. Analytical problem over motion of elliptic cylinder under free surface by Kostikov and Makarenko [10] determines asymmetric wave patterns generated on free surface caused due to rotation. And also transversely oscillating cylinder in a uniform flow by Zheng and Zhang [11] investigates frequency effects on flow induced wake for both near and away vortex shedding natural frequency.

In this paper, the flow over a cylindrical body is concentrated at Re $5 \times 10^3$ and $10 \times 10^3$, which corresponds to the intermediate subcritical regime. Over this range of Reynolds number, transition in free shear layer seek dominance over the wake and transition eddies are formed as a chain along free shear layer and precede the transformation to turbulence. Simulation is performed on a two dimensional mesh configuration of the cylinder with varying shapes having major concern for vortex shedding analysis. On basis of literature review [12] which reveals considerable flow distortion for blockage ranging from 6% to 16% compared to that of unblockage. The result presented in this paper is considered for three blockage 6%, 10% and 40% at $5 \times 10^3$ Re. The analysis over blockage change is extended to shape change for a particular blockage 6%, by variation in major and minor axis ratio through three different- 0.75, 1 and 1.5 aspect ratio considerations at $10 \times 10^5$ Re. The flow analysis behind the bluff body is covered by collection of pressure intensity, velocity intensity in two dimensions and turbulent
kinetic energy intensity data up till a range of 3 times the cylinder diameter behind the cylinder in both dimensions for blockage variation and 3 times the cylinder radius for aspect ratio variation.

2. Geometrical configuration and mathematical formulation

The aim here is to simulate fluid flow over a cylinder in 2-Dimensional configurations at the centre axis of a confined surrounding Figure.1. Three blockage, as pre-defined on the basis of previous studied Norberg [13]. Three blockage ratio 6 %, 10 % and 40 % of the flow region is used to determine the diameters of the circular cross section of the cylinder. For 10 % Blockage diameter (= 6 mm) is calculated using Equation. (1).

\[ 1 - \frac{(60 \times 60) - (60 \times d)}{60 \times 60} = 10\% \]

(1)

Similar calculation for 6 % and 40 % blockage give 3.6 mm and 24 mm diameter values respectively.

![Figure 1. Schematic a flow past a circular cylinder (all dimensions are mm).](image)

For each blockage ratio three different shapes are subjected to fluid flow analysis. The aspect ratio of shape of bluff body is governed by three different major to minor axis ratio 0.75, 1 and 1.5. Inlet velocity for simulation run of each case is obtained in respect of Reynolds number 5000 by Equation. (2).

\[ Re = \frac{L \times v}{\nu} \]

(2)

Where, \( Re \) is the Reynolds number, \( L \) is the characteristics length \( v \) is the inlet flow velocity \( \nu \) is the kinematic viscosity. For case of 10 % 0 % and 6 % blockage and \( Re = 5000 \) the velocity values are 0.833 m/s, 0.416 and 1.388 m/s respectively due to increase the size of bluff body while keeping total computational domain constant. The two-dimensional mesh for various cases is modeled in ICEM Ansys software. O-grid specification mesh is formed in cylinder vicinity for refinement in simulation outcomes. Further, unstructured mesh are exported for each case and simulated in ANSYS-Fluent.
In the present simulation, flow past circular cylinder has been computed by applying the following boundary conditions as follows:
(a) Inlet uniform flow \( (U = 1.0, \: V = 0.0) \)
(b) Cylinder surface - No slip \( (U = 0.0, \: V = 0.0) \)
(c) Top and Bottom Boundaries symmetry boundary condition.
(d) Outlet Boundary - continuative boundary condition can be expressed as \( (P = 0.0) \)

**Figure 2.** Unstructured mesh for 10 \% blockage

### 3. Validation

A grid independent mesh structure is designed for the advantage of comparison with published results of researches and to serve as reliable information source for future research purposes. Mesh number density were tested to ensure that the solution was independent of the mesh. These meshes along with drag and lift coefficient prediction are reported in Table 2 for grid independent study. In addition, the plot of \( C_D \) and \( C_L \) is also presented in Figure. 3 against number of nodes in mesh. A mesh with minimum number of nodes, after which values of coefficient of drag and lift do not change as observed from Figure. 3 and is selected as grid independent mesh. It is established that a mesh with 31700 nodes is optimum for simulation. All the above cited mesh structures consist of same number of nodes.

**Table 1.** Effect of number of nodes on drag and lift coefficient at \( Re = 5000 \) and time steps is 2500 to 3500

| Number of nodes | \( C_D \) | \( C_L \) |
|-----------------|----------|----------|
| 35220           | 1.480763 | 1.07589  |
| 31700           | 1.465363 | 1.06451  |
| 24118           | 0.834492 | 0.028372 |
Figure 3. Plot of drag and lift coefficient with number of nodes

4. Result and Discussion

4.1 Variation of drag coefficient

Figure 4. Variation of pressure coefficient for different aspect ratio corresponding to 6% blockage at $Re = 5 \times 10^3$
Figure 4 represents the variation of pressure coefficient with respect to different shapes for 6% blockage condition. Evidently minimum value of pressure coefficient is resulted in the case of aspect ratio 0.75, and is in ascending order for aspect ratio 1 and 1.5. An inverse relation of coefficient of drag with aspect ratio is indicated by the observation which is explained in the following paragraph.

Figure 5 represents the variation of drag coefficient with blockage ratio at different aspect ratio (major to minor axis ratio). It is observed from the Figure 5 that the drag coefficient gradually increases with increase in blockage ratio at aspect ratio 0.75. However, as the aspect ratio increases to 1.5, the drag coefficient remains unaltered with small variation. This variation in drag coefficient profile indicates, the effect aspect ratio is more compared to blockage on the circular cylinder. Further, at aspect ratio one, the drag coefficient is constant at initial blockage and thereafter it $C_D$ increases linearly with further increase in blockage ratio. This is similar observation reported by West and Apelt for circular cylinder [12].

![Figure 5. Variation of drag coefficient with different blockage and aspect ratio at Re $5 \times 10^3$.](image-url)
Figure 6. Represents pressure contour of cylinder with blockage: 6%, 10%, and 40% from left to right respectively at Re $5 \times 10^3$.

Figure 7. Represents turbulent kinetic energy contour for cylinder with blockage: 6%, 10%, and 40% from left to right respectively at Re $5 \times 10^3$. 
Figure 8. Represents pressure contour for cylinder with blockage 6% and varying aspect ratio as 0.75, 1.00 and 1.50 from left to right respectively at Re $10 \times 10^3$.

Figure 9. Represents turbulence kinetic energy contour for cylinder with blockage 6% and varying aspect ratio as 0.75, 1.00 and 1.50 from left to right respectively at Re $10 \times 10^3$.

The pressure contour for different blockage ratio is shown in Figure 6. It is found that the lower pressure which corrected with the wake zone of bluff body gradually reduces with increase in blockage. This is explained with the Figure 7 which is contour of turbulent kinetic energy. This turbulent kinetic energy examines the contribution of flow mechanisms such as convection, production, diffusion and dissipation in the TKE transport equation [14]. This is found here that wake zone is reducing with reduction in turbulent kinetic energy. This shows a strong correlation between pressure and turbulent kinetic energy. The pressure and turbulent kinetic energy contour is shown for different aspect ratio in Figure 8 and
Figure 9. It is found that aspect ratio 0.75 the wake zone increased and similarly turbulent kinetic energy is also high. The wake zone is gradually reducing as move to the different aspect ratio. From these figures, it is clear that the pressure redistribute the mean flow turbulent kinetic energy and except fluctuations, it can be captured with RANS simulations.

5. Conclusion

The 2D numerical simulation is done to predict the effect of blockage and aspect ratio of bluff body. The grid independent and validation is done and found well matching. The pressure and turbulent kinetic energy contours shown for different blockage and aspect ratio. It was found that pressure is well correlated with mean turbulent kinetic energy and it redistributes the flow energy.

6. References

[1] Gera B, Sharma P K and Singh R K 2010. International journal of applied engineering research, Dindigul 1(3) 602-610.
[2] Dey P and Das A K R 2017. Indian academy of science, Sadhana 42 1155–1165.
[3] Zdravkovich, M M 1997. Oxford University Press 1.
[4] Doolan C J 2010. Journal of Engineering Applications of Computational Fluid Mechanics 4 496-510.
[5] Morkovin 1964. ASME Symposium on fully separated flows 102-118.
[6] Roshko A 1961. Journal of Fluid Mechanics 10 (3) 345-356.
[7] Labraga L, Kahissim G, Keirsbulck L and Beaubert F 2007. Journal of Fluids Engineering 129 1203-1211.
[8] Miozzi M, Capone A, Felice F D, Klein C and Liu T 2016. PHYSICS OF FLUIDS 28, 124101.
[9] Jelita M, Mudia H and Afriani S 2017. Advances in Theoretical and Applied Mechanics 10 11 – 20.
[10] Kostikov V K and Makarenko N I 2016. Journal of Physics: Conference series 722 012021.
[11] Zheng Z C and Zhang N2008. Journal of Fluids and Structures 24 382-399.
[12] West, G S and Apelt C J 1982. Journal of Fluid Mechanics 114 301-377.
[13] Norberg C N 2006. Cambridge University Press 258 287-316.
[14] Liu X and Thomas F O 2004. Experiments in Fluids 37 469–482