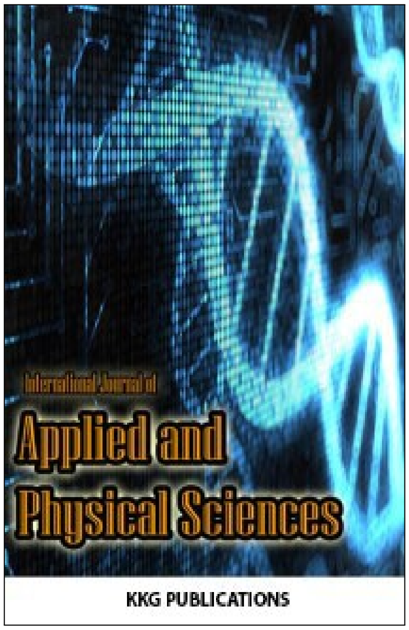

Key Knowledge Generation
Publication details, including instructions for author and subscription information:
http://kkgpublications.com/applied-sciences/

Numerical Investigation of the Flow Structure around a Cylinder for Different Fluids and Reynolds Numbers

PERIHAN OCAL [1], KAZIM PIHTILI [2]

[1, 2] Faculty of Engineering, Bingol University, Bingol, Turkey

Published online: 20 November 2017

To cite this article: P. Ocal and K. Pihtili, "Numerical investigation of the flow structure around a cylinder for different fluids and Reynolds numbers," *International Journal of Applied and Physical Sciences,* vol. 3, no. 3, pp. 60-67, 2017.
DOI: https://dx.doi.org/10.20469/ijaps.3.50002-3

To link to this article: https://kkgpublications.com/wp-content/uploads/2018/09/IJAPS.3.50002-3.pdf

KKG PUBLICATIONS

International Journal of Applied and Physical Sciences
2017, 3(3): 60-67

IJAPS

NUMERICAL INVESTIGATION OF THE FLOW STRUCTURE AROUND A CYLINDER FOR DIFFERENT FLUIDS AND REYNOLDS NUMBERS

PERIHAN OCAL [1]*, KAZIM PIHTILI [2]

[1, 2] Faculty of Engineering, Bingol University, Bingol, Turkey

Keywords:
Steady State Flow Around a Cylinder
Laminar Flow Around a Cylinder
Flow Around a Blunt Body
Drag Coefficient

Received: 03 March 2017
Accepted: 10 August 2017
Published: 20 November 2017

Abstract. In engineering areas such as chimneys, bridges, nuclear power plants, airplanes, submarines, and high buildings are exposed to fluid or gas flows. As we know, there are vibrations, loads, vortexes around bodies that are exposed flows. Also, these undesirable situations have required some solutions. In this regard, for solutions to such problems, analysis of flow around bodies concerning velocity distribution, streamlines, and vortexes are significant. From this perspective, because of its easy geometry, the flow around the cylinder has been investigated on velocity distribution, streamlines, vortexes for different fluids, and Reynolds numbers through Analysis CFX 14.5. The laminar and steady flow was handled for two different cases. Case 1 and case 2 differ in terms of boundary conditions for related fluids and Reynolds numbers. Finally, from the results in question vortex zones, both cases' velocity and pressure distribution, and drag coefficient are compared for case 1 and case 2.

INTRODUCTION

Understanding flow structure around blunt bodies is an important phenomenon due to come up with solutions for problems related flow physics. For examples blunt bodies exposed to air or water flow is pipe lines, risers, submarines, bridges feet, chimneys, airway vehicles etc. Flow structure also can be changed with a lot of variables such as Reynolds number, different fluid, velocity structure of flow, boundary conditions, velocity boundary layer, cross flow effect, vicinity of wall near to the blunt body etc. Change in any of these variables will be changed the characteristic of flow effect around body. This study handled to understand different boundary condition effect on flow characteristic over smooth cylinder with respect to wide range different Reynolds numbers and different fluids for steady laminar flow. So from results velocity distribution, pressure distribution, wake region vorticity, drag coefficient versus with related Reynolds numbers were investigated.

There are a lot of literatures in this theme because there is a lot of variables effect the characteristic of flow over blunt bodies. For example in [1] numerical investigation on three dimensional flow over cylinder was taken into consideration for steady and unsteady state for Reynolds number equal to 39000. In study in question firstly steady state handled, then unsteady state handled then two different states were compared with each other. For both states boundary conditions were taken as velocity inlet, pressure outlet, symmetry and periodic interface. For domain dimensions in Cartesian coordinate X, Y, Z was 32D, 16D, 4D respectively. Unsteady state investigated for two different cases, for case 2a the velocity profile taken as u(y) do not uniform but for case 2b uniform velocity boundary condition applied. Wall the vicinity of the cylinder taken into consideration to understand wall effect on flow characteristic. It was observed that velocity profile for wake region agreement with previous studies. It was observed that for case 2a and case 2b vortex shedding was suppressed. In [2] numerical investigation of unsteady flow around a cylinder was discussed. To observed wake dynamic and flow structure unsteady flow over rotating cylinder taken under microscope in related study. Finally it was observed that circumferential velocity exceeding about two times flow velocity the von-Karman vortex shedding occurred for low rotational speed disappeared.

In [3], [4], [5] the modeling of flow around a cylinder and analyzing and swirl control was examined. Controller design has been done for swirl measurement by means of MATLAB based Navier 2d. Then for certain conditions analyses has been done. With help of the swirl controller the vortices occurred behind the cylinder have begun to be suppressed by blowing air at the top and bottom of the cylinder. In another study [6] flow around a circular cylinder is investigated by using Abacus/CFD. In this study CFD analysis accomplished for flow over a circular cylinder exposed to uniform velocity.

*Corresponding author: Perihan Ocal
†Email: pocal@bingol.edu.tr

From results various occurrences point out related with von-Karman vortexes. It was observed that for $R > 200$ transition flow occur from laminar to turbulent and so three dimensional vortexes occurs. As it is known drag coefficient most important parameter in respect to flow around blunt bodies. To evaluate drag coefficient in [7], [8] experiments on flow over a circular cylinder accomplished for different cylinder diameters as 12.5 mm, 15 mm, 25 mm and different air velocities. For calculation of drag coefficient two main methods were used which known as direct weighing method and pressure distribution method. Calculated coefficients compared with each other in terms of two methods. Important deviation occurred for two methods. Of course it was observed weighing method was higher accurate rate than pressure distribution method.

PHYSICAL MODEL AND BOUNDARIES

Physical model is represented in Figure 1. For both cases, Figure 1 (a) represented case 1; Figure 1 (b) represented case 2. Case 1 and case 2 differ from one other in term of boundary conditions which illustrated related Figures. So the effect of the boundary conditions on the flow was tried to be understood. Model in question is taken into consideration in 3D dimension (X, Y, Z) in dimensionless form. These dimensions were taken as 32D, 16D, 0.25D respectively for both cases. The center of the cylinder is placed 8D away from inlet and 8D away from top and bottom boundaries.

In present study cylinder dimension D is equal to 2 mm. Also other dimensions were taken as accordance with this magnitude. Doman inlet was taken as uniform velocity distribution and velocity magnitude was calculated accordance with related Reynolds number and related fluid properties of course. Temperature of fluid was taken as 25°C for both fluids. As seen from physical model Figure 1 at outlet boundary condition relative pressure magnitude was taken as zero Pa. Top and bottom edges were taken as no slip walls and adiabatic walls so shear stress in related region was considered as zero. At front and back edges of channel, symmetry boundary conditions were considered.

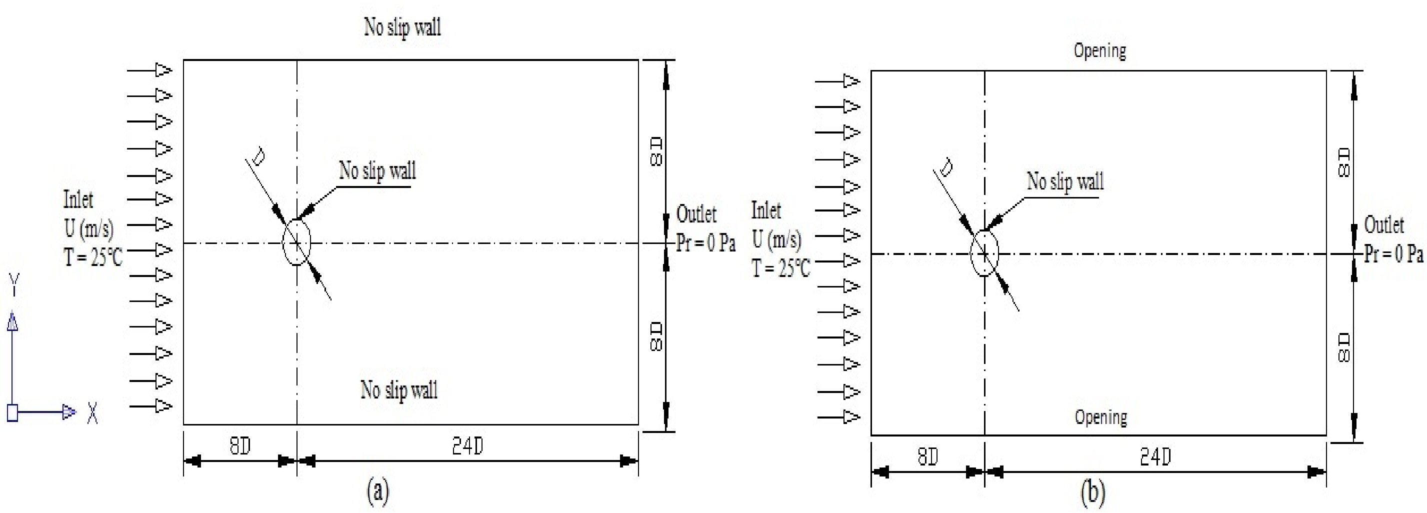

Fig. 1 . Physical model, a) Case 1 model, b) Case 2 model

Two different fluids were used for flow domain, which consists of air and water.

Mesh configuration and mesh accuracy shown in Figure 2 (a), and Figure 2 (b) respectively. Mesh accuracy was proved for related element sizes which demonstrated in Figure 2(b). As seen from that Figure three different mesh sizes were compared to ensure accuracy in respect to centerline velocity gradient. 345393, 510625, 821171 element sizes were taken into consideration. That is to say for three different mesh sizes accuracy was proved about rate of 100%. All numerical investigations were handled for element sizes 821171.

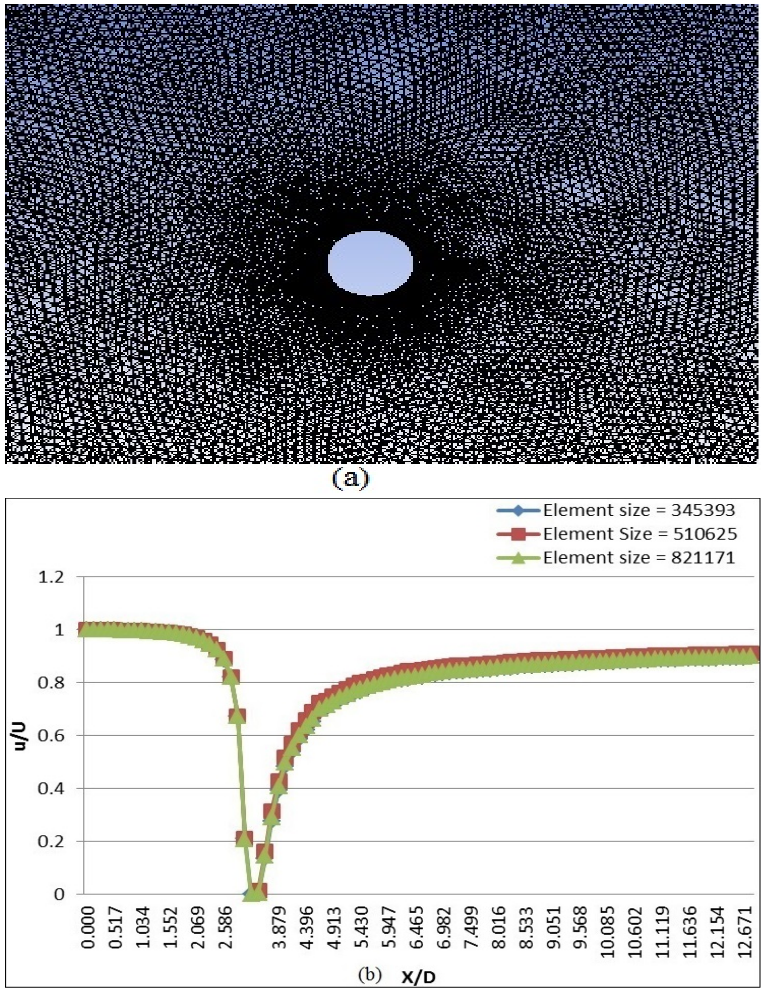

Fig. 2 . Mesh structure, a) Mesh configuration, b) Mesh accuracy

Numerical simulation done by analysis CFX solver is based on RANS form of momentum and mass conservation equations. RANS form of momentum and mass conservation equations for incompressible flows using by CFX fluid domain solver with Cartesians coordinate system shown in equation (1) and (2).

$$\rho\left(u_i\frac{\partial u_j}{\partial x_i}\right)=\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_i}\left((\mu+\mu_i)\frac{\partial u_j}{\partial x_i}\right)\qquad j=1,2,3 \quad (1)$$

$$\frac{\partial u_i}{\partial x_i}=0 \qquad (2)$$

To come up with suitable comments related results, drag coefficient must be calculated. As known drag coefficient is depend on drag force magnitude, flow velocity, fluid density, and related area. That is to say it can be formulated like equation (3).

$$C_D=F_D/(1/2.d.U^2.A) \qquad (3)$$

RESULTS AND DISCUSSION

Velocity Distribution

Figure 3 (a) and Figure 3 (b) show velocity distribution of case 1 and case 2 respectively. Because of the similarity of velocity distributions for both flow related Figures shown for water flow only. As it is seen for case 1 the wall boundary condition effected the velocity distribution because of boundary layer. As we know when a flow contact with wall, its velocity takes its zero value at the wall, its' value increases as farther from wall. Like that from Figure 3 (a), wall effect is seen which demonstrated that velocity magnitude change from zero at wall to its local value at further region. For low Reynolds Number boundary layer thicker than high Reynolds numbers. This indicates that the frictional forces at this Reynolds number are more dominant than the inertial forces on the wall. As we know Reynolds number refers to the ratio of the inertia forces to the frictional forces. Wall effect observed up to Re 100 for case 1, of course for case 2 there is no wall effect, opening boundary effect can be observed. If the Reynolds number is greater than 100 for case 1, the enhanced flow was observed. When Reynolds number equal to 200 the velocity distribution of case 1 and case 2 are close to each other (Figure 3(a) and Figure 3(b)). Velocity distribution of case 2 takes its enhanced form nearly at Reynolds number equal to 100. That is to say at case 2 flow take its enhanced form for lower Reynolds number then case 1 due to the wall effect. A significant point is that for case 1 velocity magnitude change from inlet to outlet because of viscous effect of wall boundary condition, but there is no significant change velocity distribution in case 2 due to the opening boundary condition. Another important situation is that for both cases velocity magnitude change due to solid cylinder wall effect.

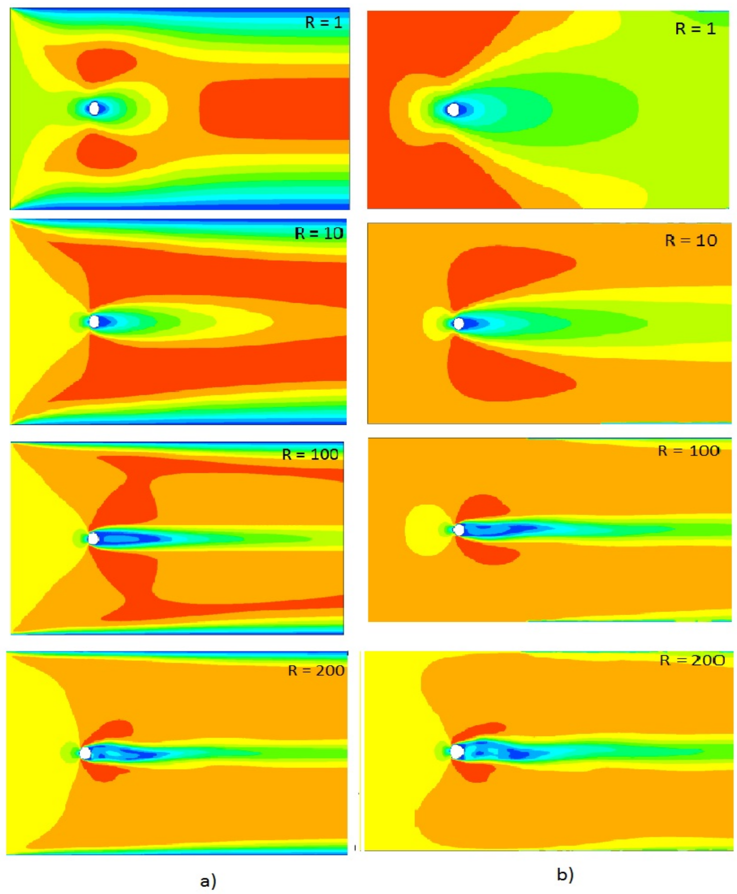

Fig. 3 . Velocity distribution of water flow a) Case 1, b) Case 2

Static pressure distribution shown at Figure 4(a) and Figure 4(b) indicated case 1 and case 2 respectively. Pressure distribution shown for water only, because of similarity was observed for both flows. From pressure and velocity Figures mainly it can be said that because of the pressure increase in front of the cylinder the velocity inclined to decreases. Especially for low Reynolds number e.g., for Reynolds equal to 1 and 10 stagnation zone occur and it cause the velocity to fall to zero. At rear side of cylinder due to the flow separation and vorticity, pressure increases, so velocity decreases. This situation can be observed almost for all Reynolds numbers in both cases. Velocity distribution differ from one another especially for low Reynolds number e.g., for Reynolds equal to 1 and 10. If the Reynolds number higher than 10 pressure distribution for both cases is resemble. In generally, it is seen that from Figure 4 at stagnation point pressure has its maximum value and it has its minimum value at rear side wall of the cylinder.

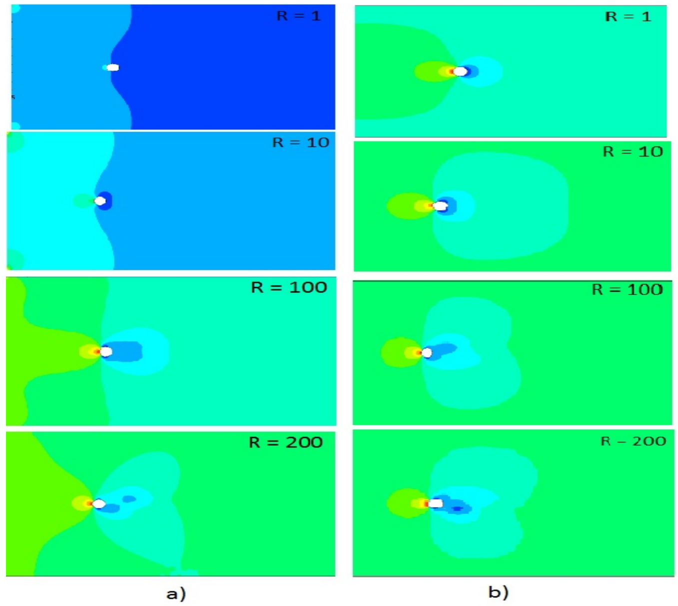

Fig. 4 . Static pressure distribution of water flow a) Case 1, b) Case 2

Understanding pressure, velocity distribution, drag coefficient better we firstly need to understand structure of separation, wake region, boundary layer, vortex shedding also. In this sense velocity streamline distribution is shown with Figure 5 which prepared for water flow only. Because of similar configuration were observed for air and water, It is not necessary to show air velocity streamlines Figure. According to both Figures 5 (a & b) for low Reynolds number, there are no vortexes at rear of the cylinder. So related flow almost can be described as potential flow. As it is known at low Reynolds number the flow is defines as potential flow so there is no vortex and slightly separation yet.

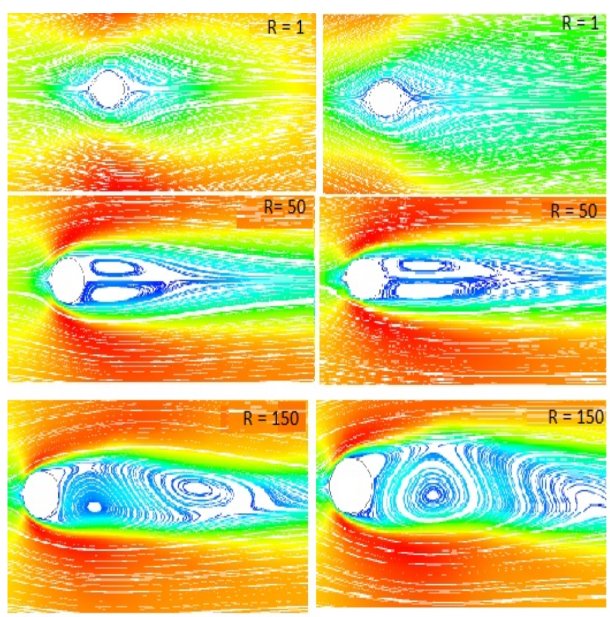

Fig. 5 . Velocity streamlines distribution of water flow a) Case 1, b) Case 2

Reynolds value ranged from 10 to 50 the separation and eddies become clear. If Reynolds number slightly greater, vorticities break off. For Reynolds number equal to 150 Von-Karman vortex shedding come into existence. In this sense in [1] it was mentioned that there is no separation for $R < 5$, but for Reynolds number from 5 to 40 there is a pair of vortices at wake region and also it was mentioned, according to reference study for Reynolds number under 189 vortex shedding is laminar and two dimensional which almost compatible with our work.

Figure 6, shows vorticity distribution for case 2 for air flow. Vorticity distribution of case 2 also similar to case 1. According to [9] for Reynolds number greater than 40 the boundary layer over the cylinder will separate due to adverse pressure gradient. Separation point where the shear stresses has its zero value. As seen from Figure separation point change with different Reynolds number. As a result of that, shear layer occur. Then the boundary layer contained amount of vorticities form around of the cylinder, then they cause shear layer to roll up in to a vortex [1]. First vortex pair at present study was observed at Reynolds number equal to 50 (Figure 5). It is significant saying that higher Reynolds number the increases length of the shear layer and also vorticity length.

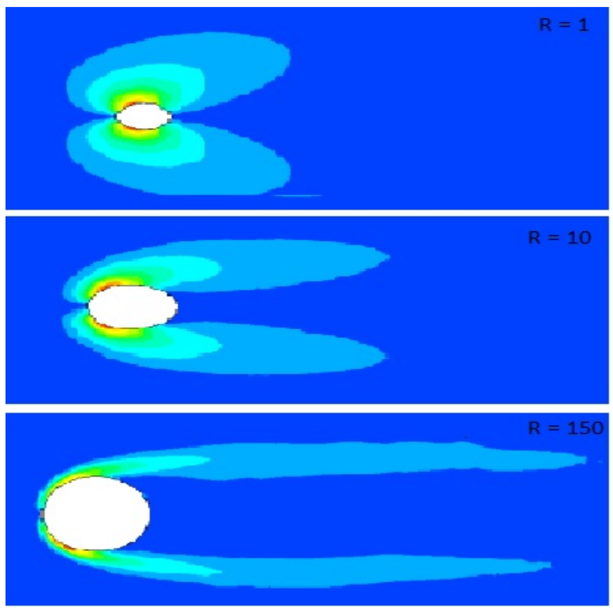

Fig. 6 . Boundary layer, shear layer, vorticities

As we know drag coefficient more important parameter for blunt bodies which contact with fluid. It is undesirable condition for most flow dynamic especially for airfoils, submarines, risers etc. In this sense in Figure 7, drag coefficient versus with Reynolds number is shown for case 1 and case 2. As seen from Figures that as Reynolds number increases the drag coefficient decreases for both cases and both fluids. This situation also has been confirmed in [10]. Another significant point may be that at Reynolds number equal to 1, case 1 drag coefficient is higher than case 2 for the same conditions. It should be emphasized that for low Reynolds numbers drag coefficient sharply decreases, on the other hand for higher Reynolds number it decreases slowly. Wall effect causes increase of drag coefficient as compared to case 2 for opening condition. That is to say as we understood from results drag coefficient is depend on Reynolds number and for higher Reynolds number the lower drag coefficient occur.

Drag coefficient also depend on boundary condition of domain in consideration, as you can see for wall vicinity to flow drag coefficient is increasing compared with flow without wall vicinity, on the other hand if there is no slip wall near the flow field drag coefficient is decreases of course. It is apparently understood that the drag coefficient of water and air flow is the same for the same Reynolds numbers as seen from Figure 7 for both cases. Apart from these, drag coefficient closely related with shape of blunt body of course, but in present study this situation is not taken into account.

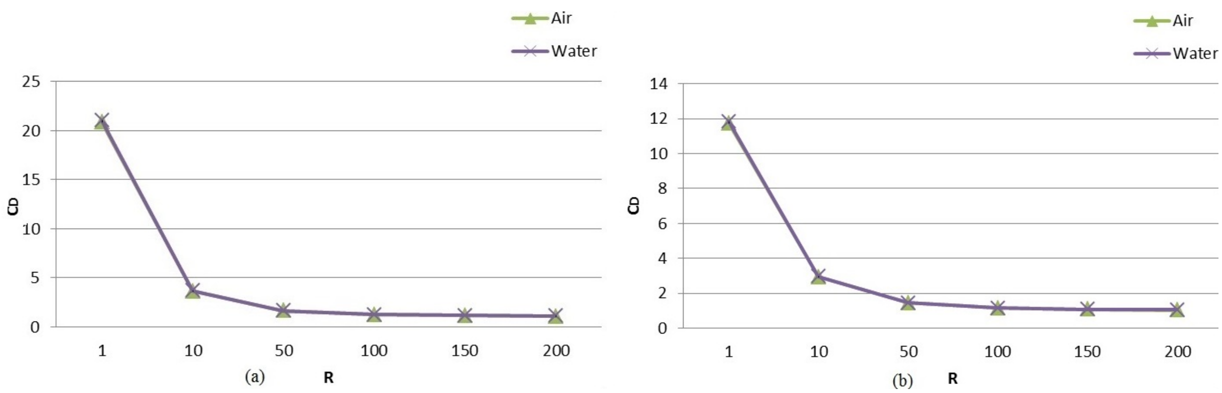

Fig. 7 . Drag coefficient versus with Reynolds number a) Case 1, b) Case 2

CONCLUSION

In this study as mentioned before for two different cases numerical analysis was done. Physical model of case 1 and case 2 illustrated by Figure 1, as seen from case 1 physical model solid cylinder placed at a channel, top and bottom of channel taken as no slip wall. It helps us to understand wall effect on flow around a cylinder. Then as seen from case 2 boundaries in question were handles as opening boundary condition which means that flow into domain possible. For boundary conditions in question, laminar steady state flow around blunt body was taken into consideration for different Reynolds number and different fluids. From results accomplished velocity distribution, pressure distribution, streamlines colored by velocity were shown; comparison of drag coefficient in terms of different boundary conditions, different fluids and different Reynolds numbers was taken under microscope. Results were important in term of velocity distribution effected by no slip wall in case 1, likewise drag coefficient results are significant of course. In the case of wall effect, drag coefficient is higher than the case of no wall effect for the same number of Reynolds. Also for the same Reynolds numbers forming of the same values of drag coefficient for both fluids is interesting. In this sense further studies can be carried out by investigating the drag coefficient for different fluids at different Reynolds numbers. Also investigation on different blunt bodies' shape, more important in terms of reducing drag coefficient. That is to say we believe that the work in question will be a good reference for future work in this field. It is possible to say briefly that:

1. Velocity distribution shows that for case 1 no slip wall boundary condition affected velocity distribution, especially make it zero at wall because of boundary layer, then of course effect all distribution from inlet to outlet. It is possible to observe the fully developed state of the velocity for a given Reynolds number, i.e., the velocity distribution at a given Reynolds number becomes similar to case 2. For Reynolds number equal to 200 the velocity profile of both cases almost fully enhanced (Figure 3).

2. In generally at in front of the body because of the stagnation zone lowest velocity value observe, at rear of the cylinder because of the separation, vorticity effect caused velocity taking low value in this region. On the other hand velocity at top and bottom of cylinder take it maximum value (Figure 3).

3. At Reynolds equal to 1 there is no any vortex. On the other hand First pair vortex occur for Reynolds number equal to 50, then these vortex break off and wash down away from rear of cylinder. Von-Karman vortex shedding occurs at Reynolds number around 150 (Figure 5).

4. Vortices Figure show that for different Reynolds number the region of separation point and length of the shear layer of course length of the vortices extended. As the number of Reynolds increases, the length of the vorticities increases (Figure 6).

5. Drag coefficient result show that when the presence of the no slip wall condition (case 1), drag coefficient value increases compared with presence of the opening boundary condition (case 2). As increases Reynolds number drag coefficient decreases. At low Reynolds number this decline is so sharply. On the other hand reduction in the drag coefficient at higher Reynolds number is slower (Figure 7).

REFERENCES

[1] M. Abrahamsen, "Simulation of viscous flow around a circular cylinder with Star-CCM+," Master dissertation, Department of Marine Technology, Norwegian University of Science and Technology, Trondheim, Norway.

[2] A. B. Bagi, "Numerical investigation of unsteady flow around circular cylinder," Unpublished thesis, Department of Mechanical Engineering, University of Washington, Seattle, WA, 2015.

[3] H. D. Karaca, G. D. Ozen and C. Kasnakoglu, *Modeling, analysis and swril control of flow around cylinder.* Brussels, Belgium: European Union, 2008.

[4] S. Wittayapiyanon and S. Chuepeng, "Sports utility vehicle seat structure simulation of frontal oblique crash," *Journal of Advances in Technology and Engineering Research,* vol. 3, no. 3, pp. 69-78, 2017.

[5] M. S. Nugroho and M. Suryanegara, "Analysis of interference of Unmanned Aircraft System (UAS) and fixed service at frequency band 12.5-12.75 GHz by considering the factor of rain attenuation," *Journal of Advances in Technology and Engineering Research,* vol. 2 , no. 5, pp. 164-169, 2016.

[6] M. Sato and T. Kobayashi, "A fundamental study of the flow past a circular cylinder using Abacus/CFD," in *SIMULIA Community Conference,* Providence, RI, 2012.

[7] M. Mallick and A. Kumar, "Study on drag coefficient for the flow past a cylinder," *International Journal of Civil Engineering Research,* vol. 5, no. 4, pp. 301-306, 2014.

[8] Sugiyatno, I. Djunaedi and N. A. Mahardiono, "Modelling and simulation of hybrid control system in solar cell-battery-super capacitor," *International Journal of Technology and Engineering Studies,* vol. 1, no. 3, pp. 74-80, 2015.

[9] F. Jorgen and B. M. Summer, *Hydrodynamics around cylindrical structures.* Singapore: World Scientific, 1997.

[10] K. Y., Ivanovich, M. Z. Myo and K. A. Yurievich, "Aerodynamics investigation for prospective spacecraft vehicle in the transional regime," *International Journal Aeronautical and Space Science,* vol. 14, no. 3, pp. 215-221, 2013.

— This article does not have any appendix. —