Numerical Investigation of 2D Turbulent Flow past a Circular Cylinder at Lower Subcritical Reynolds Number

K M Sowoud¹, A A AL-Filfily¹ and B H. Abed¹
¹Department of Power Mechanics Engineering, Engineering Technical College-Baghdad, Middle Technical University, Baghdad, Iraq

Abstract. The main objective of the present study is to evaluate the applicability of the standard k-ε turbulence model in engineering practice in the subcritical flow regimes. Also, examine the influence of Reynolds number variation on the flow characteristics are considered. The numerically simulation of the flow characteristics over a smooth uniform cross section circular cylinder with a diameter of 50 mm and 400 mm span, placed horizontally perpendicular to the main flow direction at four different Reynolds numbers (Re) based on the cylinder diameter (D) and the undisturbed free-stream velocity in the range \( R_e = (0.5, 0.7, 0.85 \text{ and } 1.0) \times 10^5 \) were performed. The simulated results of the flow properties such as flow stream velocity components, pressure distribution contours, pressure coefficient \((C_p)\) and drag coefficient \((C_D)\) around the circular cylinder were consider. The numerical analysis obtained by CFD using ANSYS FLUNET (19.1) with K-ε turbulence model, for solving the 2D (ANSE) governing equation for tested model were consider. The present simulation results validation is obtained by comparing the converged results with published numerical and experimental data.

Keywords: Circular cylinder, Drag force, subcritical Reynolds number, ANSYS FLUENT.

Nomenclatures

- \( D \): Drag force; \( D = \frac{1}{2} \rho V_\infty^2 AC_D \)
- \( C_D \): Drag coefficient; \( C_D = \frac{2F_D}{\rho V_\infty^2 A} \)
- \( C_p \): Pressure coefficient; \( C_p = \frac{P-P_\infty}{q_\infty} \)
- \( q_\infty \): Free stream dynamic pressure; \( q_\infty = \frac{1}{2} \rho V_\infty^2 \)
- \( F_D \): Drag force
- \( V_\infty \): Free stream velocity
- \( \mu_\infty \): Free-stream air dynamic viscosity = 1.81 x 10^-5 Pa.s
- \( d \): Diameter of the circular Cylinder
- \( b_2 \): Cylinder length
- \( A \): Projected area; \( A = b_2 \times \pi D \)
- \( \rho_\infty \): Free-stream air density = 1.225 (kg/m3)
- \( P \): Local static pressure on cylinder periphery
P<sub>∞</sub>: Static pressure in free stream
CD: Drag coefficient of cylinder

1. Introduction

Flow around a circular cylinder is essential for researchers and engineering design aspects. Moreover, the flow over the circular cylinders have a wide range of applications such as in heat exchanger, cooling towers, nuclear cooling system, bridge piers, subsea pipelines can be modeled as cylinder. Most of engineering problems are subjected to sub-critical Reynolds number. Boundary layer separation and flow oscillation in the wake zone behind the cylinder occur due to shedding of vortices at moderate and high Reynolds number. These types of flow can sometimes results to undesirable structural vibrations, which leads to structural damage. Therefore, this kind of flow behavior is very interesting to be studied in deeply especially the effects of the vortex shedding on the cylinder.

For real fluid the flow pattern is more complicated than that for inviscid fluid and varies depending upon the Reynolds number, which is represent the ratio between the inertia force (Fi) and viscous force (Fv). Figure 1 depicts a variety of flow patterns as the velocity (Reynolds number) of the fluid increased.

At low Reynolds number (Re<1), no separation occurs, viscous forces are predominated and the flow may be generally consider laminar. The flow pattern is smoothly in a symmetric not only around its front and rear, but also around the upper and lower side and thereby no wake formed, as shown in Figure 1a. The drag force is relatively high, as shown in Figure 2. A change take place in the flow patterns as a Reynolds number increased slightly at about 4< Re <40. The flow separates on the downstream side, and symmetric pair of fixed vortices are formed in the near weak (Figure 1b). These vortices are stable and remain attached to the body.

As the Reynolds number increased about 40< Re <100, the new flow pattern were develop. The wake behind the circular cylinder become unstable and one of the two vortices in Figure 1b are break away and then the second is shed because of the flow oscillation and the nonsymmetrical pressure in the wake zone, as shown in Figure 1c. This phenomenon is known as Karman Vortex Street. In the Reynolds number range 100 < Re < 200, the vortices become unstable, as shown in Figure 1d. Further increase in Reynolds number 200 <Re<400, periodic irregular disturbances are start in the wake. The flow is transitional and gradually becomes turbulent, as shown in Figure 1e. As the Reynolds number is increased 400 < Re <2.5 × 105, the boundary layers separates at about 800 on the front half of the cylinder, as shown in Figure 1f. This flow pattern causes the pressure in the wake zone is much lower than the free-stream pressure. This flow is called subcritical flow.
Figure 1. Regimes of fluid flow over circular cylinder [1].

When the Re is increased in range $2.5 \times 10^5 < Re < 3.5 \times 10^5$, the flow come into the critical limits and some references called it transitional flow. The boundary layer separation itself becomes turbulent and unsteady. A dramatic reduction occurs in drag over 70% [1], as shown in Figure 2. With further increasing in Reynolds number $3.5 \times 10^5 < Re < 3 \times 10^6$, the separated region becomes turbulent, reattaches, and separates again at $120^\circ$ on the rear back of the cylinder, as shown in Figure 1g. The drag force again increased, as shown in Figure 2 [1]. When the Reynolds number become Re$>3 \times 10^6$, the boundary layer become turbulent on the front side of the cylinder and the separation take place at $140^\circ$, as shown in Figure 1h. The pressure is somewhat lower resulting in drag increased.

Figure 2. Variation in the drag coefficient with various Reynolds numbers [1].

Extensive experimental and numerical efforts have been devoted to understand the flow characteristics (wake zone) downstream the cylinder at both laminar and turbulent flow conditions. Majtaba Daneshi [3], studied the flow characteristics such as velocity fields, pressure distribution and drag force for unsteady incompressible turbulent and laminar flow downstream a circular cylinder numerically and analyzed by using ANSYS Fluent software and 2D finite volume method. Also, the effect of Reynolds number Re $10^5$ and $10^5$ on flow fields were computed. The influence of aspect ratios (L/D)(diameter...
7.5 and 10cm, length 65cm) on flow parameters are investigate experimentally by using sub-sonic wind tunnel and numerically by using ANSYS (14.5) software by Kartik Chandra Bhagat et al. [4]. Results show that the boundary layer separation takes place as the result of pressure distribution over the cylinder of an angle 90°-100°, also was found that the error percentage 13.75% between the experimental and numerical methods.

Mehmet I. Yuce et al. [5], studied numerically the flow characteristics around circular cylinder for laminar (Re=2) and turbulent flow (Re=4×10^5). 2D simulation using K-Ε turbulence model were carried out. The result obtains shows that the wake zone behind the square cylinder much more than that behind the circular cylinder. the flow in the wake zone of different profiles of bluff bodies such as triangle, square and circular at Re=5000 and 10000 are investigate experimentally and numerically by S. Yagmur et al. [6]. Results showed that a good agreement in respect of flow patterns of vortices, velocity and streamline topology. M. M. Rahman et al. [7], studied the wake zone downstream the circular cylinder for incompressible, unsteady, laminar and turbulent 2D flow. 2D finite volume method is using to solve the governing equations. The drag and pressure coefficients for different Reynolds numbers (Re=10^3 and 3.9×10^3) are computed and compared with the other numerical data. Their results show that a good agreement with the experimental results.

CFD, numerical simulation using finite volume method to study the (2D, incompressible and unsteady) flow behavior over a square cylinder were carried out by Gera.B, et al [8]. Also, investigate numerically the wake zone downstream the cylinder for Reynolds number range 50-250. Their results show that the vortex shedding start between Re=50 and Re=55, the effect of Re in drag, lift and strouhal number were predicted and shows a good trend with the other work results. Jie Shao at el. [9], Re-average Navier-Stokes equations and large eddies simulate (LES), these two methods are used to simulate the 2D flow over a circular cylinder at Re=5800. For 2D flow the turbulent K-Ε model is used. By comparing the results with experimental data are found that the 3D (LES) is more necessary to solve this complex flow more than 2D (RANS) equation.

Study the flow behavior numerically at the wake zone of the bluff body at different blockage ratios and Reynolds numbers by Pankaj Kumar at el. [10], Computational Fluid Dynamic (CFD) with ANSYS/ Fluent using a finite volume method are considered. The drag, pressure coefficient and kinetic energy variation are computed and analyzed. The result shows that the drag coefficient increased with increasing blockage ratio at fixed Re while reduced with increase Re at fixed BR. In this research, flow around a smooth circular cylinder had been numerically investigated. Since, both the geometry of circular cylinder and free-stream velocity had been chosen to achieve the subcritical Reynolds number (300 < Re < 2×105). In the literature, the subcritical flow past circular cylinder has received much less attention compared with flow at lower Reynolds number Re < 300 and supercritical flow Re>3.5 × 106. The main goal of the present research is to investigate numerically the behavior of fluid flow in the wake zone of a smooth uniform cross section circular cylinder with a diameter of 50 mm and 400mm span by using the CFD, ANSYS/ Fluent with K-Ε turbulence model for solving the 2D (ANSE) governing equations. Numerical simulation of velocity (contours, vectors and streamlines), pressure distribution and drag coefficient are calculated and simulated at different Reynolds numbers.

2. Methodology
Methodologies of the present research start first, with the flow past the cylinder is modeled in 2D placed horizontally perpendicular to the main flow direction. The simulated model having (D=5cm) and a span 40 cm was used. The cylinder dimensions and velocities are achieved to satisfy the desirable lower subcritical Reynolds numbers. The construction of three dimensional model using ANSYS software, fine mesh generation using 3D element, the computational domain and boundary conditions used are given in Figure 1 [2].
3. NUMERICAL DESIGNS AND MODELING

3.1 Mathematical models

The flow past a smooth circular cylinder has been simulated by solving numerically and 2D Navier-Stokes equations (for unsteady, incompressible and turbulent flow). In order to solve the Navier-Stokes equations of motion (Eq.1) and continuity equation (Eq.2), the finite volume method is used in which the general transport equation for transient and 3-D flow boundary condition is given by [6]

\[ \frac{\partial \vec{u}_i}{\partial x_i} = 0 \]  
\[ \frac{\partial \vec{u}_i}{\partial t} + \frac{\partial \vec{u}_i \vec{u}_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial \{(u_i \vec{u}_j - \vec{u}_i \vec{u}_j)\}}{\partial x_i} + \nu \frac{\partial^2 \vec{u}_i}{\partial x_i \partial x_j} \]  

Where:
- \( x_i \) = The Cartesian coordinates in x directions
- \( t \) = time
- \( u_i \) = velocity components
- \( \nu \) = kinematic viscosity

In order to solve the continuity equation (Eq.1) and the Navier-Stokes equations of motion (Eq.2), the finite volume method is used in which the general transport equation for transient and 3-D flow boundary condition is given by [6].

\[ \int_C \left[ (\int_A^{t+\Delta t} \frac{\partial \phi}{\partial t} + \int_A^{t+\Delta t} n(\rho \phi) dA) dt + \int_A^{t+\Delta t} n(\nabla \phi) dA dt + \int_C \nabla \phi dV \right] dt \]

Where:
- \( \phi \) = general property and growth rate
- \( \Gamma \) = coefficient of diffusion

For the purpose of calculate the drag coefficient, the force in the X-direction has found from the analysis which is nothing but the drag force. Upon substituting the drag force value in the following equation, the drag coefficient (\( C_D \)) value can be found.

The drag coefficient is computed using equation (4):

\[ C_D = \frac{2FD}{\rho \infty AxV^2} \]  
\[ Re = \frac{\rho \infty V_{\infty} d}{\mu_{\infty}} \]

3.2 Mesh and Grid

Figure 3 shows the diagram of the present model which include the cylinder and the enclosed wind tunnel, it has cleared that the density of the mesh near the cylinder wall is condensed and consist of 100 layers of uniform elements in order to obtain a precise results.

The boundary conditions used in the model are as follow, see Figure 4:
1. Inlet, \( v = 15, 20, 25, 30 \) m/s
2. Cylinder wall, stationary wall, no slip shear condition
3. Stationary wall, no slip shear conditions
4. Outlet pressure condition, pressure gage = 0 MPa.

**Figure 3.** Shows the mesh model and the 100 layers of elements near the cylinder wall.

**Figure 4.** Boundary condition of the present work.

4. RESULTS AND DISCUSSION
Numerical analysis of the flow past a circular cylinder have been carried out using commercial CFD, ANSYS/Fluent (19.1) to solving the governing equations with the corresponding boundary conditions. The simulated results obtain for the 2D flow past a smooth circular cylinder with definite geometry (diameter=5cm and length = 40 cm) model and at four different Reynolds number (\( Re = 0.5, 0.7, 0.85 \) and 1.0)\( \times 10^5 \) depend on the free stream velocity. In this work, all the results data for drag and pressure coefficients are calculated and compare to each others.
Figure 5. Flow stream velocity component contours (left side) and velocity vector contours (right side) for circular cylinder at different Reynolds number.

Computed the stream velocity component contours (right side) and velocity vectors (left side) are presented at Reynolds numbers ($Re = 0.5, 0.7, 0.85$ and $1.0 \times 10^5$) shown in Figure 5. It is clearly that in Figure 5a to 5d, the high velocity occurs at the upper and lower sides are displayed by red contours on the circular cylinder. Whereas, the blue contours at the front and rear represent the lower velocity contours, separation of boundary layers occurs at an angle about $80^\circ$ [1].

In Figure 6a to 6d, shows the velocity streamlines distribution around the circular cylinder with different velocities. The pressure distribution around the tested model (circular cylinder) at four different Reynolds numbers, are shown in Figure 6e to 6h. A remarkable difference in pressure distribution can be seen. It clearly observe that the front face of the cylinder is exerted to high pressure ($+C_p$) which is specified by red area. This can be interpreted due to the flow slowdown the front face of the cylinder and become stagnation region. The flow again accelerated at the upper and lower sides of the cylinder causes pressure to drop and resulting in low pressure distribution ($-C_p$) which is indicated by blue area. Turbulent kinetic energy contours around the circular cylinder with different Reynolds numbers are shown in Figure 7.
Figure 6. Flow streamline contours (left side) and pressure distribution contours (right side) at different Reynolds numbers (Re=0.5, 0.7, 0.85 and 1.0) ×10^5.
Figure 7. Turbulent kinetic energy contours around circular cylinder at different Reynolds numbers.
Figure 8. Pressure coefficient variation as function of angle around circular cylinder.

The calculated numerically results of the drag coefficient ($C_D$) and drag force for circular cylinder at each Reynolds numbers ($Re= 0.5, 0.7,0.85$ and $1.0) \times 10^5$ are presented in Table 1.

The drag coefficient ($C_D$) variation with Reynolds numbers for circular cylinder is plotted in Figure 9. It is evident that the trend of the drag coefficient is increased as the Reynolds numbers increased. The maximum drag coefficient ($C_D=0.089$) occurs at high subcritical Reynolds numbers $1.0 \times 10^5$ as shown in Figure 9. This can be interpreted due to separated boundary layers causes the turbulent flow downstream the circular cylinder (wake zone) with full in vorticties.

Figure 9. Drag coefficient ($C_D$) variation with Reynolds numbers.
Table 1. Drag coefficient ($C_D$) and drag force for circular cylinder at different Reynolds numbers.

| Speed (m/s) | Re       | CD  | $F_D$ (N) |
|-------------|----------|-----|-----------|
| 15          | $0.5 \times 10^5$ | 0.028 | 3.91      |
| 20          | $0.7 \times 10^5$ | 0.042 | 5.82      |
| 25          | $0.85 \times 10^5$ | 0.062 | 8.58      |
| 30          | $1.0 \times 10^5$ | 0.089 | 12.35     |

5. Conclusion:
In present research, the wake zone of a smooth circular cylinder with identical geometry placed horizontally perpendicular to the free-stream flow at different Reynolds number were studied numerically. In addition, the pressure coefficient ($C_p$) and drag coefficient ($C_D$) of the cylinder were also calculated. For numerically study ANSYS-Fluent with K-ε turbulence model were used.

The following conclusion may conclude from the present study:
1. From the numerical simulation results, the wake zone length downstream the cylinder at $Re = 1 \times 10^5$ is longer than that at $Re = (0.5, 0.7$ and $0.85) \times 10^5$.
2. The separation of the boundary layers occurred as the results of distributed pressure past the cylinder was found at an angle ranged between $80^\circ$ to $100^\circ$.
3. It is observed that the drag force depend strongly on the Reynolds numbers. The drag force increased as the Re increased.

References:
[1] Roland L 2013 “Incompressible Flow”, Fourth edition.
[2] FLUENT Inc. ANSYS FLUENT 19.1 2018 Theory Guide.
[3] Mojtaba D, 2016, Numerical Investigation of the Fluid Flow around and Past a Circular Cylinder by ANSYS Simulation, International Journal of Advanced Science and Technology 92 49-58.
[4] Kartik C, Subrato K and Sunil K 2016 Experimental and Numerical Analysis of Different Aerodynamic Properties of Circular Cylinder. International Research Journal of Engineering and Technology (IRJET) 3(9) 1112-17.
[5] Mehmet I and Dalshad A 2016 A Numerical Analysis of Fluid Flow Around Circular and Square Cylinders Journal American Water Works Association, http://dx.doi.org/10.5942/jawwa.2016.108.0141, (E546-E554).
[6] Yagmur S,Dogan S, Canli E, Muharrem H and Ozgoren M 2015 Experimental and Numerical Investigation of Flow Structures around Cylindrical Bluff Bodies, owned by the authors, published by EDP Sciences, EPJ Web of Conferences, 92, 02113.
[7] M M Rahman, M M Karim and M. Alim 2007 Numerical Investigation of Unsteady Flow Past a Circular Cylinder Using 2D Finite Volume Method Journal of Naval Architecture and Marine Engineering, 4 27-42.
[8] Gera B, Pavan K and Singh R 2010 CFD analysis of 2D unsteady flow around square cylinder International Journal of Applied Engineering Research, 1(3) 602-610.
[9] Jie S and Chao Z 2006 Numerical analysis of the flow around a circular cylinder using RANS and LES International Journal of Computational Fluid Dynamics 20(5) 301–307.
[10] Pankaj K and Santosh S 2019 Flow past a bluff body subjected to lower subcritical Reynolds number, https://doi.org/10.1016/j.joes. (2019).