International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249-8958 (Online), Volume-8 Issue-3, February 2019

Effect of Plain Shear Velocity Profile over a Streamline Cylinder

Manish Kumar Rawat, Sanjeev Kumar Gupta

Abstract**: *Flow with plain shear velocity profile with high Reynolds Number (Re = 0.5 ×10⁶ to 3.6×10⁶) around a flat (2D) streamlined cylinder, is simulated by using a k- ε turbulence model. The main target of this study is to assess the influence of plain shear velocity and axis ratio on the important flow variables, such as drag force, point of separation and mean drag coefficient, along the exterior surface of a flat elliptical cylinder. Drag force over a circular cylinder as compare to elliptical cylinder is higher. Reduction in drag may be increased by reducing the axis ratio with plain Shear velocity profile. The point of separation moves forward as the cylinder became streamlined.***

Index Terms**: *Flat Streamlined Cylinder, Plain Shear Velocity, Axis Ratio, Turbulent Flow etc.***

I. INTRODUCTION

The practical study of flow, which is very turbulent, is very costly, tedious and inaccurate. So the majority of studies on unsteady flow, taken place over a circular cylinder, used numerical simulation and virtual environment. But in real time applications, flow over complex bodies like rotor blades, wings of plane, submarines and rockets are involved. Modeling of such type of objects cannot be possible as a flow over a symmetrical round shaped cylinder. In such flow condition, axis ratio can influence the point of separation, shape of wake region, intensity of turbulence, lift coefficient and drag force. Further, the results of turbulent flow with plain shear velocity attract interest due to its importance in the actual engineering problems. A plain shear velocity flow may influence the separation angle, separation point, pressure on the back side and the aerodynamic forces acting on it. The study of variation in coefficient of drag, separation angle and back pressure recovery for such type of complex flow is very important for the designing of real world engineering application. The plain shear velocity is very important in the study of the flows in open channel and closed channel flow in which our parabolic cylinder is near to the surface on one side while the other side is very far. There are very few simulations in the open literature on the flow of fluid having plain shear velocity profile over an elliptical cylinder. Yoshihiro mochimaru [4] numerically examines the effect of axis ratio on the various parameters and flow streamlines for the uniform velocity up to Re = 10^5. The present model was first validated by comparing the present result with the experimental results of Achenbach [1] and numerical results of Catalano et al. [3] for a circular shaped cylinder.

Then the parametric study is accomplished to evaluate of the effect of variation in axis ratios and plain shear velocity profile for the same mass flow rate on the coefficient of drag, pressure coefficient and point of separation over an elliptical cylinder, using standard k-ε turbulence model.

Manuscript published on 28 February 2019.
* Correspondence Author (s)
Manish Kumar Rawat*, Department of Mechanical Engineering, GLA University, Mathura, UP, INDIA
Sanjeev Kumar Gupta, Department of Mechanical Engineering, GLA University, Mathura, UP, INDIA

II. MATHEMATICAL FORMULATION

The Reynolds-averaged equations for conservation of mass and momentum are given by

$$\frac{du_i}{dx_i} = 0 \qquad (1)$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\left(\frac{\partial p}{\partial x_i}\right) + \nu \frac{\partial^2 u_i}{\partial x_j^2} - \frac{\partial \overline{u_i' u_j'}}{\partial x_j} \qquad (2)$$

Where i, j =1, 2. Here u_1 and u_2 are the mean velocity components in the horizontal (x_1) and vertical (x_2) directions respectively.

$\overline{u_i' u_j'}$, is the Reynolds stress component, is expressed in terms of a turbulent viscosity ν_T and the mean flow gradients using the Boussinesq approximation,

$$-\overline{u_i' u_j'} = \nu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3} k \delta_{ij} \qquad (3)$$

Where k denotes the turbulent kinetic energy and δ_{ij} denotes the kronecker delta function.

The turbulent kinetic energy (k) and its dissipation rate (ε) of k–ε turbulence model are given by:

$$\frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\nu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right) + \nu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j} - \varepsilon \qquad (4)$$

$$\frac{\partial \varepsilon}{\partial t} + u_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\nu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right) + C_1 \frac{\varepsilon}{k} \nu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j} - C_2 \qquad (5)$$

Where $\nu_T = C_\mu(k^2/\varepsilon)$.

III. NUMERICAL SOLUTION PROCEDURE

Commercial code GAMBIT is used for modeling of the flow situation and FLUENT is used to carry out the numerical simulations. It is based on a second order finite volume discretization and SIMPLE algorithm is used as a technique of pressure correction.

The eddy viscosity is calculated by standard k–ε turbulence model which is further used to calculate Reynolds stress. Unstructured mesh of C type with 400 grid points along the outer periphery of the cylinder and 100 grid points in the perpendicular direction of the surface to model the flow situation. The grid is non-uniform. The size of the cells is very small close to the external surface of the cylinder as compare to the size of the cells far away from the cylinder's boundary.

The size of the computational domain in the flow direction is 27D while in the normal direction 14 times of diameter is used is used for the present simulations. The far field effect is mitigated by placing the flow inlet at 7D upstream and flow outlet at 20D, towards trailing edge, from the centre of the cylinder.

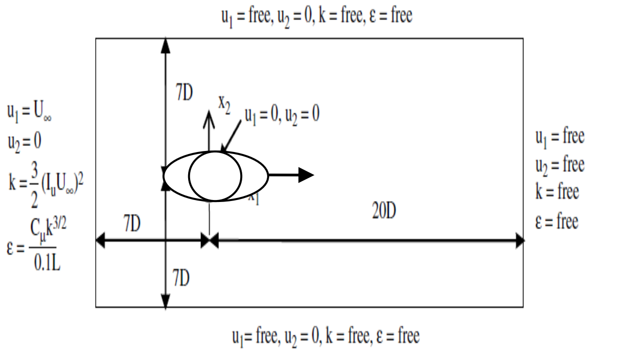

Fig.1: Computational domain

IV. RESULTS

The computations have been performed in the highly turbulent flow regime where the Reynolds number varies between 0.5 ×106 and 3.6×106. The estimation of the variation in the point of separation, drag force and skin friction drag over the peripheral surface of the streamlined cylinder with axis ratio for plain shear velocity input is the main objective of this study.

First the numerical approach is validated with the results of previous researchers for a circular cylinder with uniform velocity at the inlet and then a parametric study is done to highlight the impact of shape variation and plain shear velocity at the inlet of computational domain.

A. Validation

For the validation of present numerical approach, the predicted average drag coefficient and distribution of pressure coefficient for a circle, at Re = 1 $\times 10^6$ are compared with the different results. Table. I shows the overall drag coefficient at Re = 1 $\times 10^6$ and fig. 2 shows the distribution of coefficient of pressure.

Table I: Coefficient of drag results for numerical and experimental studies.

Re = 1× 10^6	C_D
Present Numerical Approach	0.40
M. C. Ong et al. [2]	0.51
Catalano P. et al. [3] URANS	0.41
Achenbach, E.	0.21–0.63

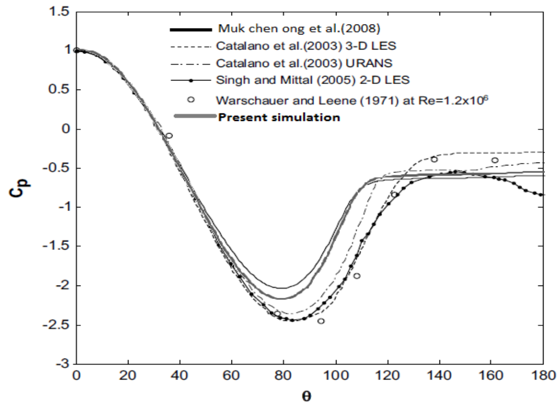

Fig.2: Coefficient of pressure variation at Re = 1×10^6

The predicted drag coefficient and distribution of pressure coefficient $C_p = (p - p_\infty)/0.5\rho U_\infty^2$ at Re = 1×106 are within the limits of the experimental result given by Achenbach, E. and showing agreement with the numerical results.

At higher Reynolds number, the pressure variation shows better agreement with experimental than at lower Reynolds number. In addition to this, The angle of separation (θs) at Re = 3.6×106 is 118o, as predicted by present simulation, is very close with the experimental result of Achenbach [1], who measured the angle of separation as 115o.

Hence the present numerical approach can be used for the simulation and design of the streamline cylinder for such complex type of flow situations.

B. Parametric Study

First, the attention is given for the estimation of the effect of shape of cylinder and plain shear velocity at the inlet on drag coefficient, pressure coefficient and point of separation. The results for the elliptical cylinder are compared with the outcomes of circular cylinder with various axis ratios at 0.5×106 ≤ Re ≤ 3.6×106 for the better designing of the streamlined surface with the required reduction in drag force and optimum strength.

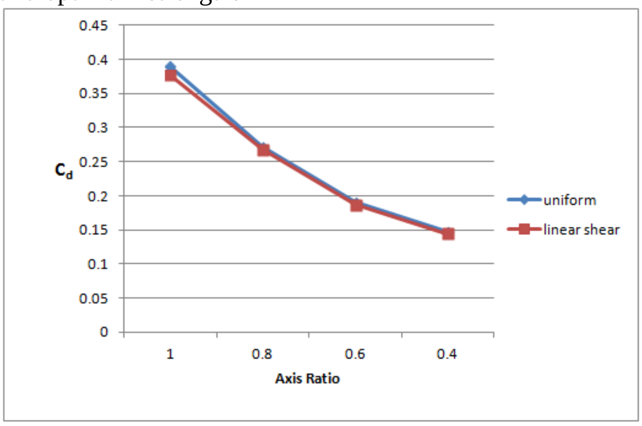

Fig. 3: Variation in coefficient of drag.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

The outcomes of the present study can be used to decide the circumstances under which the streamlined cylinder is feasible to reduce the drag force without compromising the required strength to sustain in the high Reynolds number flow which can produce a large amount of stress. Drag force for streamline body can be reduced significantly as compare to bluff body like circular cylinder for uniform as well as plain shear velocity profile. For example, CD at λo = 0.4 is 50 % less than that at λo = 0.8 and 60% less than the round shape cylinder. For any type of cylinder, a small difference is observed in the coefficient of drag for uniform velocity and plain shear velocity. Coefficient of drag for plain shear velocity profile is little higher as compared to uniform velocity profile.

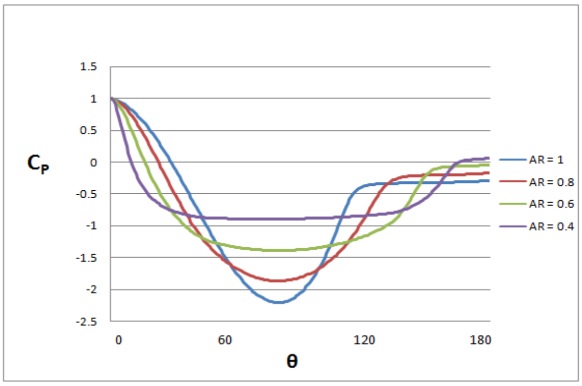

Fig.4: Coefficient of pressure distribution with axis ratios

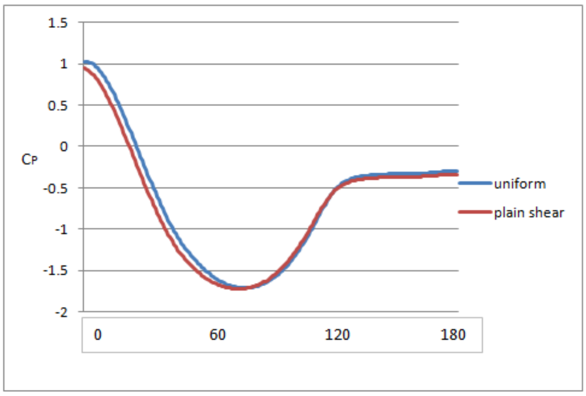

Fig.5: Coefficient of pressure variation with velocity profiles

The pressure coefficient variation over the outer peripheral surface of a streamlined body having various axis ratios is shown in fig. 4. As the body became more streamlined, a recovery is observed in the back side of of the object. Back pressure coefficient is positive for elliptical cylinders for axis ratio less than one while it is negative for circular cylinder. Uniform pressure distribution can be obtained at very low value of axis ratios. Hence the low value of drag force can be achieved. Fig. 5 displays the comparison of uniform velocity with plain shear velocity profile on the variation of the coefficient of pressure over an elliptical cylinder. The lower coefficient of pressure at the backside is predicted for uniform velocity profile than plain shear velocity profile.

The variation in angle of separation with axis ratio, velocity profile and Reynolds number is shown in table II. The separation point moves towards downstream side as the cylinder became streamlined. A small increment in the value of separation point is also observed in case of plain shear velocity profile. Due to this, the pressure drag is reduced on the cylinder with reduced axis ratio and plain shear velocity profile.

Table II Separation angle for uniform and plain shear velocity profile at $0.5 \times 10^6 \leq Re \leq 3.6 \times 10^6$

Velocity profile	λo	Angle of Separation (Degree)
	0.4	157-162[1]
Uniform velocity	0.6	138-147
	0.8	121-133
	1.0	110-118
	0.4	158-163
Shear velocity	0.6	139-149
	0.8	121-134
	1.0	111-119

The separation angle slightly increases as Re increased. For example, for an ellipse with λo = 0.4, flow separates at 157° at $Re = 0.5 \times 10^6$ and 162° at $Re = 3.6 \times 10^6$.

V. CONCLUSION

The numerical study over an elliptical cylinder with plain shear velocity indicates the reduction in drag force as the flow separation takes place on the downstream side of the cylinder due to streamlining of the cylinder. The drag force for circular cylinder is approximately 50 % higher as compare to a streamlined cylinder with λo = 0.6 for uniform as well as for plain shear velocity profile. The phenomenon of back pressure recovery and shifting of point of separation to the trailing edge, with decreasing axis ratios, is very important to understand the variation in the coefficients of aerodynamics (Cd and Cl). Due to recovery in the pressure at back side and shifting of point of separation towards trailing edge, pressure drag reduce and skin a friction drag increase which is responsible for the reduction in total drag force. All the predictions indicate the increment in separation angle with the slenderization of the cylinder. Overall the present numerical approach is very precious and helpful for complex flow situations but in the absence of experimental, some more systematic investigation with other models like LES is required for the validation of the numerical methodology.

REFERENCES

1. Achenbach, E., 1968. Distribution of local pressure and skin friction around a circular cylinder in cross-flow up to $Re= 5\times10^6$. J Fluid Mech; 34(4):625–39.
2. Ong, MC., Utnes, T., Holmedal, LE., Myrhaug, D., Pettersen, B., 2009. Numerical simulation of flow around a smooth circular cylinder at very high Reynolds numbers. Marine Structures 22: 142–153.

Published By:
Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP)

3. Catalano, P., Wang, M., Laccarino, G., Moin, P., 2003. Numerical simulation of the flow around a circular cylinder at high Reynolds numbers. International Journal of Heat and Fluid Flow 24: 463–469.
4. Mochimaru, Y., 1992. Numerical simulation of flow past an elliptical cylinder at moderate and high Reynolds numbers, using a spectral method. 11th Australasian fluid mechanics conference.
5. Li, Z., Davidson, J., Mantell, S., 2005. Numerical simulation of flow field and heat transfer of streamlined cylinders in crossflow. ASME Summer Heat Transfer Conference, 2005
6. Mittal, R., Balachandar, S., 1996. Direct Numerical Simulation of Flow Past Elliptic Cylinders. Journal of computational physics 124**,** 351–367.
7. Singh, SP., Mittal, S., 2005. Flow past a cylinder: shear layer instability and drag crisis. Int J Numer Meth Fluids 2005,75–98.
8. Tutar, M., Holdo, AE., 2001. Computational modeling of flow around a circular cylinder in sub-critical flow regime with various turbulence models. Int J Numer Meth Fluids 2001;35, 763–84.
9. Rodi, W., 1993. Turbulence models and their application in hydraulics. A state-of-the-art review. IAHR Monograph Series. 3rd ed. Rotterdam, Netherlands: A.A. Balkema; 1993.[10] Launder, BE., Spalding, DB., 1972. Mathematical models of turbulence. London: Academic Press.

AUTHORS PROFILE

Manish Kumar Rawat , M.Tech in Thermal Engineering, Assistant Professor in the department of mechanical engineering, GLA University, Mathura from 2013. He is also a member and faculty coordinator of the Indian Society of Heating, Refrigerating and Air conditioning for Engineers. He is working on Computational Fluid Dynamics and published 4 research paper in peer reviewed journals and conferences.

Sanjeev Kumar Gupta is currently working as an Assistant Professor in the Department of Mechanical Engineering, GLA University Mathura. He has more than 8 year of teaching experience. He has completed M.Tech in Fluids Engineering from MNNIT, Allahabad. His research area is Thermal Fluid engineering. He has published 10 research papers in various national and international journals of repute. . He is life member of National Society of Fluid Mechanics and Fluid Power and Indian Society of Theoretical and Applied Mechanics.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)