CFD Analyses of Two Dimensional Laminar Flow Past an Ellipse of Given Aspect Ratio

Pankaj Sharma[1], Manjunatha K R[2]
[1, 2] Dept of Mechanical
[1, 2] Nitte Meenakshi institute of technology, Bangalore-560064

Abstract- *The flow around an ellipse opens up the way to study flow past arbitrary shaped bodies which often manifests itself as some combinations of the flow past simple-geometry bodies such as cylinders or ellipses. The objective of the present study is to compute laminar flow around ellipses of varied aspect ratios and to validate the same against some reliable measurement data reported in literature. Drag force experienced by flow past ellipses have important practical impact in applications. Ellipse being a simple geometry, may be explored easily for both computational and experimental investigations the study on flow past ellipse opens up the way of studying further more complex shapes like hulls of the ships and submarines. The present work focuses mainly on laminar flow past ellipses with four different aspect ratio and each for four different Reynolds number. Important results on the flow have also been compared to corresponding measurement data reported in literature.*

Keywords- Laminar flow, Elliptic Cylinder, Aspect Ratio, Reynolds number, Wall Shear Stress, Pressure Coefficient

I. INTRODUCTION

In order to understand the 3D flow past a Prolate Spheroid, the basic knowledge about laminar two dimensional flow about an ellipse of given aspect ratio is of prime importance. At low Reynolds number the flow is likely to be laminar and steady but as the Reynolds number is increased, the flow tends to become unsteady and eventually turbulent. The present work investigates the flow past an ellipse of aspect ratios ranging from 1 to 6. At high values of Reynolds number, the flow becomes unsteady leading to flow separation and vortex shedding which may also be predicted by the present RANS3D code using structured C grid configuration.

II. OBJECTIVE

Generation of appropriate body-fitted curvilinear grids around ellipse of varied aspect ratio using the grid-generation code MESHGEN.

The simulation of 2D laminar flow past an ellipse of a given aspect ratio at different Reynolds Number has been carried out using the in-house developed FORTRAN code RANS3D

III. METHODOLOGY

The governing equations for steady two-dimensional incompressible laminar flow in general non-orthogonal curvilinear.

$$\frac{\partial(\rho\varphi)}{\partial t} + \frac{\partial(C_{i\varphi})}{\partial x_i} + \frac{\partial(D_{i\varphi})}{\partial x_i} = JS_\varphi$$

coordinates using cartesian velocity components, may be written as the following generalized transport equation. for a passive scalar :

$C_{i\varphi}$ = convective component

$D_{i\varphi}$ = diffusive component

S_φ = source component

Control Volume & Variable Arrangement

In any finite volume method, the governing differential equations for flow are to be integrated over a finite number of control volumes covering the computation domain formed by the grid generation procedure. All the variables are stored at the geometric center P of the control volume. The six neighboring control volume centers are denoted by N, S, E, W, T and B for the north, south,west, east, top and bottom neighbors respectively.

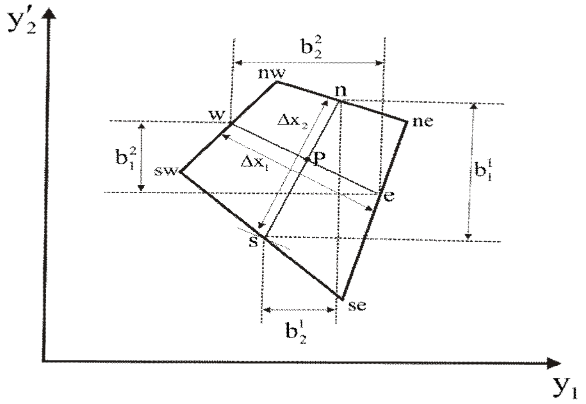


Fig. 2D Control Volume showing the relevant projection areas

Finite Volume Equation

The convective ($C_{i\varphi}$) and diffusive ($D_{i\varphi}$) fluxes computed according to the flux discretization scheme and lead to the conservation of fluxes in the form of a quasilinear equation. The source terms for the steady state equations are linearized as:

$$\int J S_{\varphi} dx_1 dx_2 dx_3 = SU^{steady} + \varphi_P SP^{steady}$$

and the final equations to be solved iteratively are (two Cartesian components of velocity) of the following form:

$$\left(\sum A_i - SP\right)\varphi_p^{n+1} = \sum_I A_i \varphi_i + SU$$

$$A_P\varphi_P = A_W\varphi_W + A_E\varphi_E + A_S\varphi_S + A_N\varphi_N + SU$$

Computation of Pressure Field

Algorithms based on pressure-velocity solution strategy transform the continuity equation into a field equation for pressure correction using momentum equations as the link between pressure and velocity corrections U'_{iw} as:

$$U_{iw} = \alpha_v\left[H_P^i + D_2^i(P_s - P_n) + D_1^i(P_W - P_P)\right] + (1-\alpha_v)U_{iw}^G$$

$$U'_{iw} = \alpha_v \overline{D_1^i}\left(p'_W - p'_P\right)$$

$$H_P^i = \left(\sum A_{nb}^i U_{inb} + SU\right)/A_P^i \quad \& \quad D_j^i = b_j^i / A_P^i$$

Where

A_{nb}^i represents the coefficients for neighboring nodes, the superscript '0 A_P^i' is the value at the previous iteration, is the value of A_P for U_i at the node P, b_j^i is the relevant projection area of the cell face, α_v is an under-relaxation parameter and the expressions with over represent the linear average of the same quantities evaluated at the cell-centers P and W adjacent to the face '*w*' on either side of the face Now the conservation of mass in a control volume may be expressed as:

$$C_e - C_w + C_n - C_s = 0$$

where the face mass flux C_w, may be written as follows using '*i*' as the summation index for two directions.

$$C_w = \rho_w b_{iw}^1 \left(U_{iw} + U'_{iw}\right)$$

Boundary Conditions

In addition to the governing equations the complete specifications of the mathematical problem requires the incorporation of proper boundary conditions for all the variables. The most frequently encountered boundary conditions and the relevant treatment required in the present procedure are described below.

Inflow: The values of all dependent variables are normally known at inflow boundaries and (4.5) prescribed accordingly.

Outflow: Similar to inflow, the profiles of the variables may also be prescribed at the outlet. However, in general, very little is known a prior about the outlet and therefore it is, in most of the cases, advisable to place the outflow boundaries far downstream with no reverse flow so that the results in the region of interest is practically insensitive to the treatment at outflow boundary.

Rigid Walls: For the rigid impermeable wall of the aerofoil or wing, the velocities at the wall and hence the convective flux across the boundary are set to zero. All kind of diffusive fluxes at the wall are also set to zero.

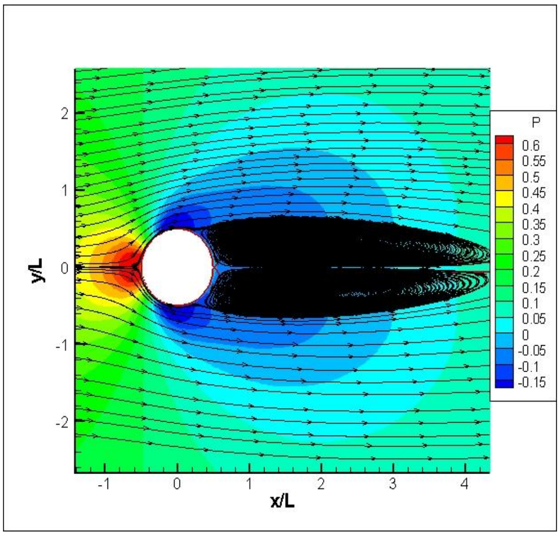
Cut Boundaries: In case of interblock cut boundaries similar to the present cut line, separating the upper and lower domain two overlapping control volumes are provided on either side of the cut boundary and the value of the flow variables in these overlapping control volumes are transferred from the running solution of the adjacent cells on the other side of the cut. Stone's[ ] strongly implicit procedure has been used consistently to solve the relevant discretised equation system.

IV. RESULTS & DISCUSSION

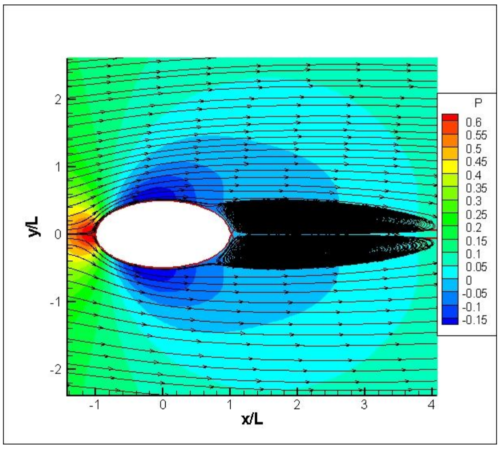
The computational results for two dimensional laminar flow past an ellipse for four different ratio between the major and minor axes (1:1, 2:1, 4:1 and 6:1)are shown below.

Flow Pattern for zero angle of attack

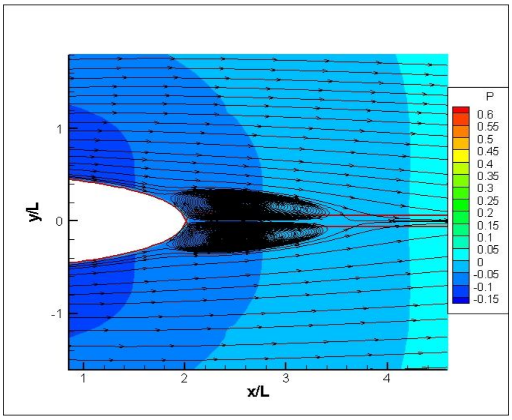
Figs below show the flow pattern around Ellipses of different Aspect Ratio at Reynolds number 100.



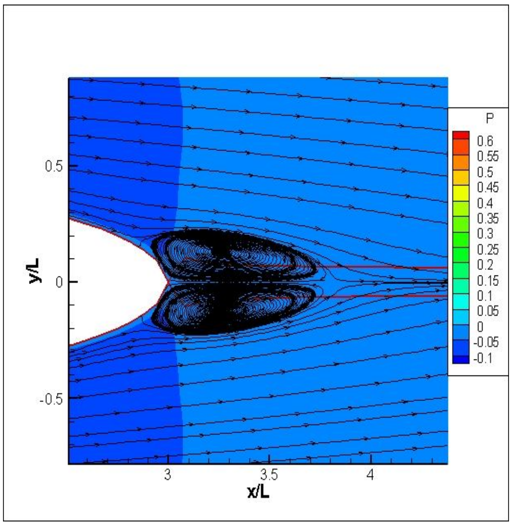
(a) 1:1 Re100



(b) 2:1 Re 100



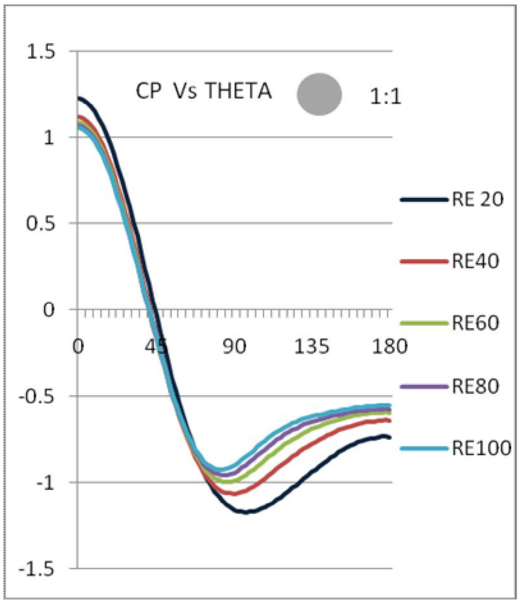
(c) 4:1 Re 100



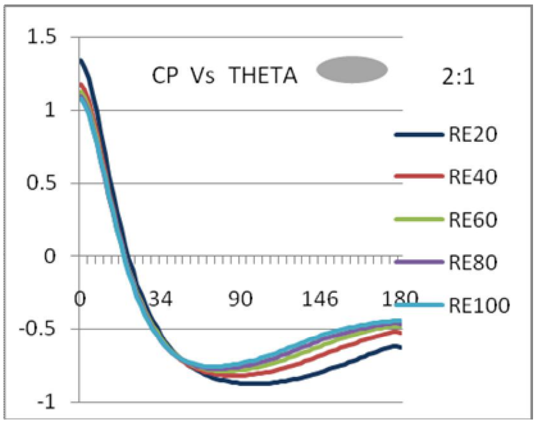
(d) 6:1 Re 100

Pressure distribution over the Ellipse

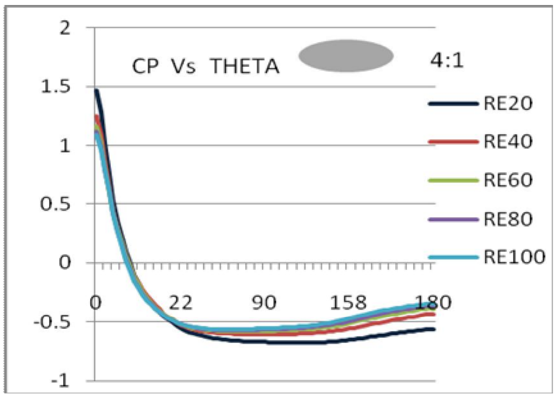
The predicted chordwise variation of surface pressure for the aerofoil at zero angles of attack for different aspect ratio are shown in below Figs.



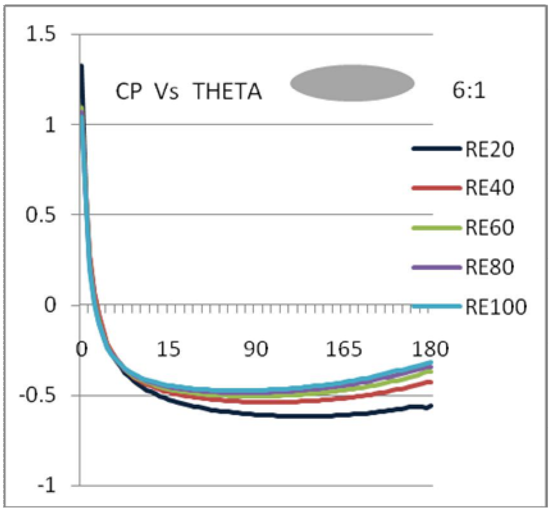
(a)



(b)



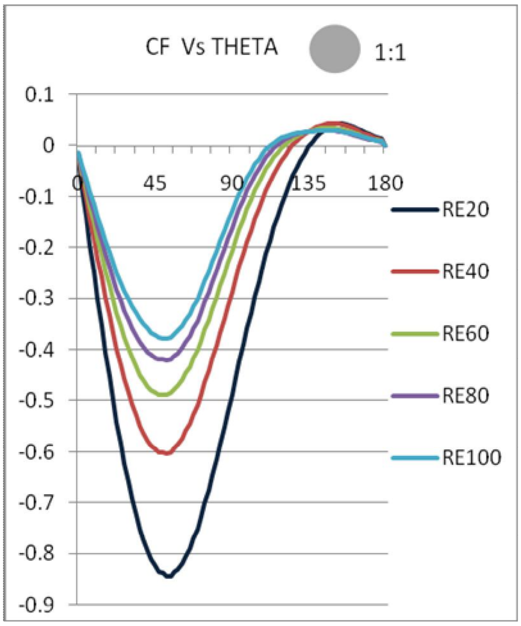
(c)



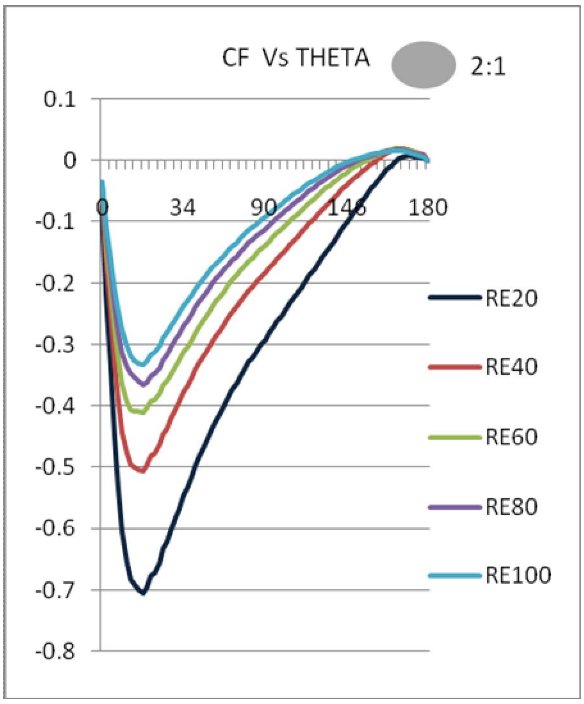
(d)

Skin Friction coefficient over the Ellipse

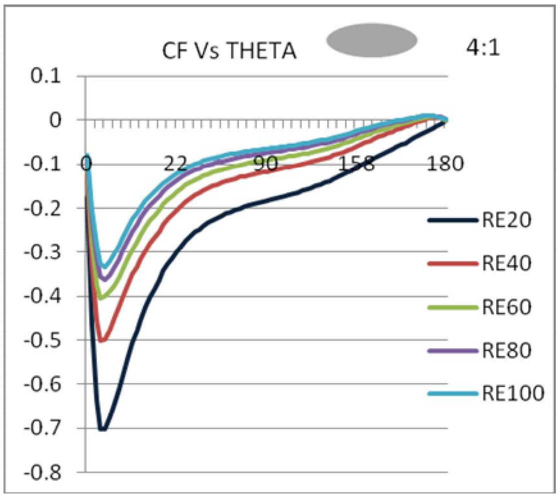
The variations of skin friction coefficient along the ellipse surface at zero angles of attack are shown in Figs below.



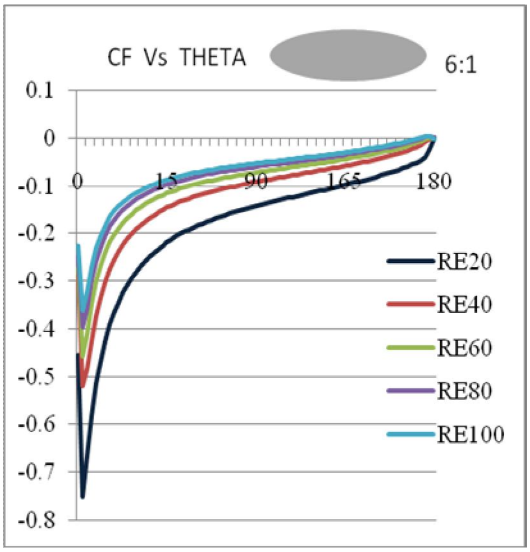
(a)



(b)



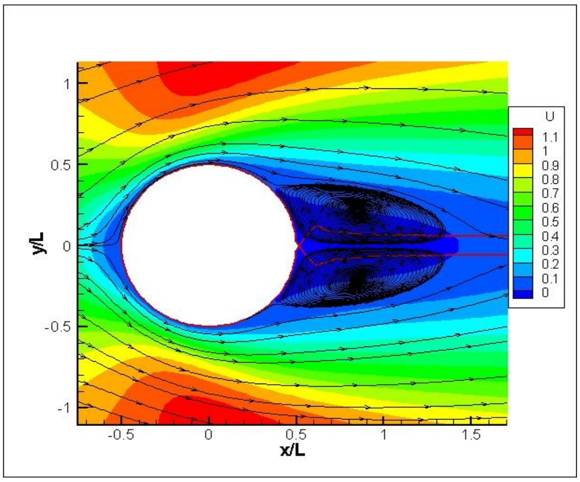
(c)



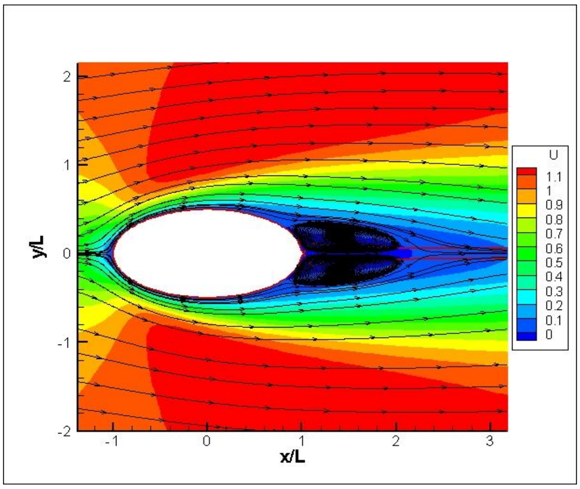
(d)

Flow pattern around the ellipse

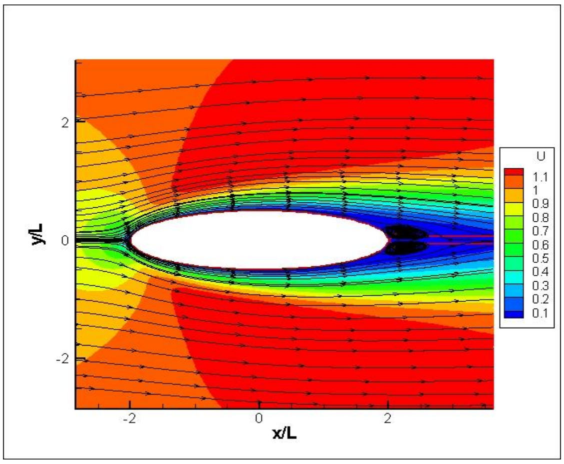
Figs shown below shows the flow pattern around the ellipse section. The pictures show the particle traces or streamlines plotted using the Tecplot360 software, based on the computed velocity vectors for the flow past ellipse at different aspect ratio.



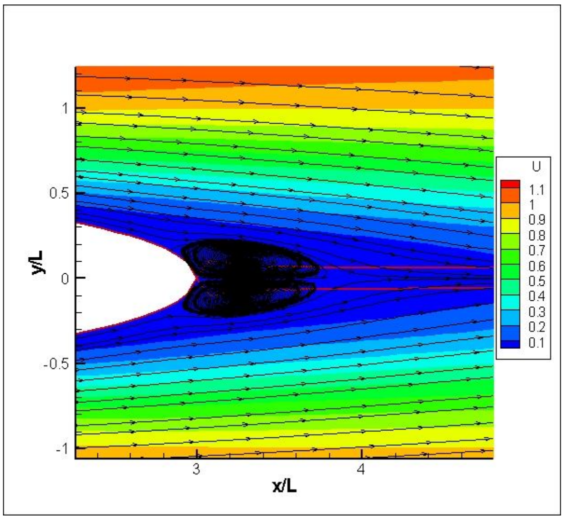
(a) 1:1 Re 20



(b) 2:1 Re 40



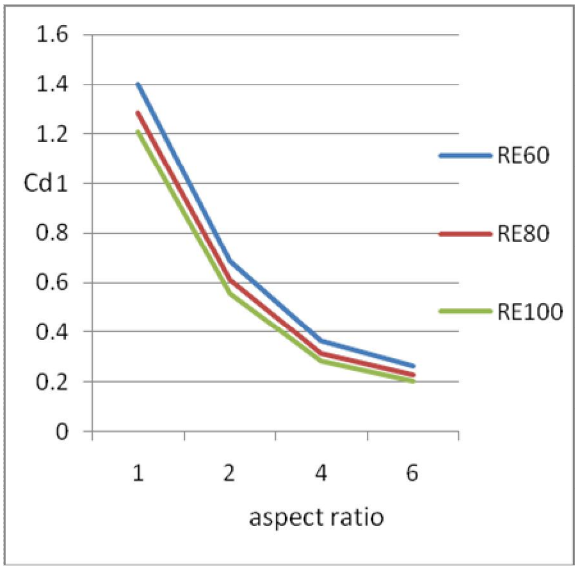
(c) 4:1 Re 60

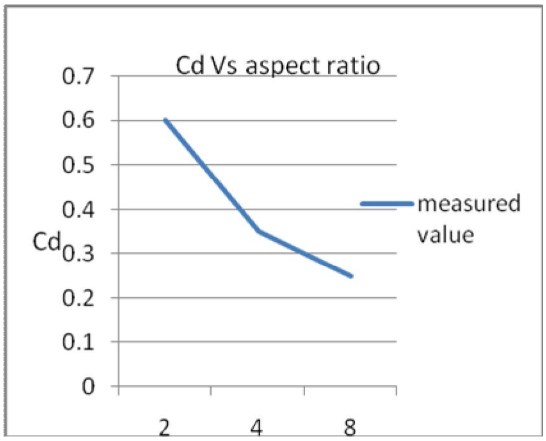


(d) 6:1 Re100

Drag for Laminar flow past Ellipses of varied Aspect Ratio

Fig shows the drag on the ellipse for all the aspect ratio (1:1, 2:1, 4:1 and 6:1) at different Reynolds number. The value of drag decreases with increase in the aspect ratio of ellipse and also decreases with increasing the aspect ratio.





V. CONCLUSION

For the same aspect ratio the coefficient of pressure decreases as the Reynolds number increases. For the same Reynolds number the Cp decreases with increase in the Aspect ratio.

For the same aspect ratio the coefficient of friction decreases as the Reynolds number increases. For the same Reynolds number the coefficient of friction decreases with increase in the Aspect ratio.

Coefficient of drag decreases as increasing the Reynolds number for same aspect ratio and the value of Cd decreases on increasing the aspect ratio.

The prediction of Cd with increase in the aspect ratio compare reasonably well with similar variation as represent in literature (Fluid mechanics by Frank M. White).

REFERENCES

[1] N. A. Meller "Viscous flow past an elliptic cylinder", Computational Mathematics and Mathematical Physics 18 (1978) 138-149

[2] S. A. Johnson, M. C. Thompson, K. Hourigan, "Flow past elliptical cylinders at low Reynolds numbers", Proceedings of 14th Australasian Fluid Mechanics Conference, Adelaide, December 10-14, 2001, pp. 343-346.

[3] S. C. R. Dennis, P. J. S. Young, "Steady flow past an elliptic cylinder inclined to the Stream" Journal of Engineering Mathematics 47 (2003) 101-120.

[4] B. M. Sumer, J. Fredsoe, "Hydrodynamics Around Cylindrical Structures", World Scientific, Singapore, 2006

[5] Z. Faruquee, D. S.-K. Ting, A. Fartaja, R. M. Barron, R. Carreveau, "The effects of axisratio on laminar fluid flow around an elliptical cylinder", International Journal of Heat andFluid Flow 28 (2007) 1178-1189

[6] Masami Sato, Takaya Kobayashi, "A fundamental study of the flow past a circular cylinder using Abaqus/CFD", Mechanical Design & Analysis Corporation, SIMULIA Community Conference, 2012

[7] Frank M. White, "Fluid mechanics"