NUMERICAL STUDY OF EXTERNAL AIR FLOW OVER STATIONARY AND ROTATING PIPE WITH ELLIPTICAL CROSS-SECTION

Maleki E. and Sadrhosseini H.*
*Author for correspondence
Science and Engineering Department,
Sharif University of Technology- International Campus,
Kish Island,
Iran,
E-mail: sadr@kish.sharif.edu

ABSTRACT

A two dimensional time dependent computational model is developed to simulate and investigate the fluid flow across a pipe with elliptical cross section for two cases: when the pipe is stationary and when it has a rotational motion. Air flow over the duct with various low Reynolds numbers (Four values in the range of Re=1 to Re=1000) is considered. Momentum equations with proper boundary conditions are solved using dynamic mesh. Streamlines, Pressure distribution and Velocity profiles are obtained and creation of vortices, boundary layers, separation region, wake region, reattachment point and stagnation points are studied in detail as well as the lift and drag coefficients as a function of time and the results are compared for the two cases.

INTRODUCTION

Fluid around a structure can significantly alter the structure’s vibrational characteristics. The presence of a quiescent fluid decreases the natural frequencies and increases the damping of the structure [1]. Fluid flow over pipes and ducts happens in many fields of engineering and has an important effect on heat transfer; such as flow over tubes in shell and tube heat exchangers, condensers and evaporators in power plants, refrigerators and air conditioners; external flow over fluid transfer pipes, or air flowing over solar vacuum tubes, as well as in aerodynamics of moving object in the air or inside the water, such as cars, planes, ships and underwater vehicles. External flow in wind turbines with various blade shapes, sizes and different angles with the wind direction affect the efficiencies of the wind power plants.

Cross flow over cylindrical structures are especially of interest in different engineering fields and industries such as offshore and civil engineering, heat exchangers design, nuclear reactor fuel rods, steel cables of suspension bridge, etc. In flow over a cylinder subjected to forced oscillations with different amplitudes, lock-on phenomenon occurs in the cases at which the vortex shedding frequency of the oscillating cylinder synchronizes with the frequency of cylinder vibrations. This phenomenon can be recognized by the spectral analysis of the lift coefficient history. At vortex lock-on region the shedding frequency is modulated to the body frequency and a single peak appears in the frequency analysis. Also the vorticity field pattern stays unchanged within the complete cycle of the cylinder vibration in the lock-on phenomenon. However, in the non-lock-on regions, the spectral analysis of the lift coefficient history indicates two distinct peaks and the vorticity field pattern changes with time within the oscillation period [2].

NOMENCLATURE

A_s		Aspect ratio
C_D		Drag coefficient
C_L		Lift coefficient
Re		Reynolds Number
t	[s]	Time
V	[m/s]	Velocity
x	[m]	Cartesian axis direction
y	[m]	Cartesian axis direction

Due to the fundamental and practical importance, the elliptical geometry has attracted many efforts. One of the primary researchers can be found in the work of Schilichting in 1968 [3]. Schinel [4] studied the effect of vortex separation on lift distribution of elliptical cylinders. Notable among the others are those by Lugst and Daube [5, 6]; they studied the flow over thin ellipse at various angle of attack for low Reynolds numbers. A comprehensive research can be found in the works of Giorgini and Avci [7] and later Hamidi and Giorgini [8] Avci and Giorgini [9] has performed a systematic study of two-dimensional flows over elliptical cylinders with various eccentricities and angel of attacks at Reynolds number ranging up to 1000. All stated studies are limited to a two dimensional flow at low to medium Reynolds numbers. In the past few years the mechanisms of flow past over an ellipse is studied by Mittal and Balachandar [10] Wang et al. [11]and Heidarnejad and Delfani [12].

In the current study a two dimensional time dependent computational model is developed to simulate and investigate the fluid flow across a pipe with elliptical cross section for two cases: when the pipe is stationary and when it has a rotational motion. Using a finite element method based on the characteristic based split (CBS) algorithm governing equations including full Navier–Stokes and continuity equations are solved. Dynamic unstructured triangular grid is used in this study employing lineal and torsional spring analogy which is coupled with the solver by an Arbitrary Lagrangian–Eulerian (ALE) formulation. After verifying the accuracy of the numerical code, simulations are conducted for the incompressible viscous flow passing across a stationary/ rotating cylinder with six different low Reynolds numbers in the range of Re=1 to 1000.

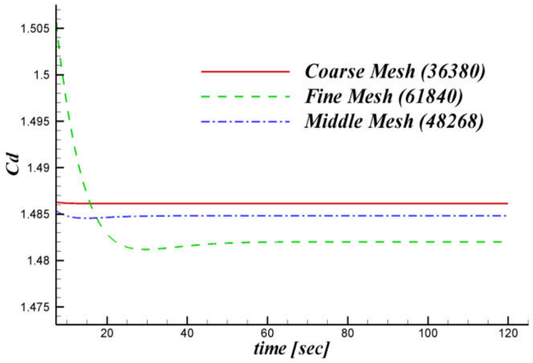
NUMERICAL MODEL AND BOUNDARY CONDITIONS

Among different numerical approaches for solving moving boundary problems, the ALE method combining the advantages of both Lagrangian and Eulerian methods employing a body-fitted moving mesh is used in this study. The computational mesh can move arbitrarily to optimize the shapes of elements, while the mesh on the boundaries precisely tracks the boundaries and interfaces. Because of this freedom in moving the computational mesh offered by the ALE description, the implementation of any kind of boundaries can be performed very easily.

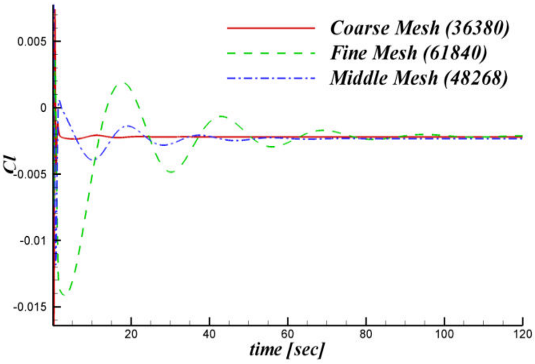
Conservation of mass and momentum are considered as the continuity and Navier Stoke's governing equations to study two dimensional incompressible cross flow over stationary/ rotational cylinder with elliptical cross section and the aspect ratio of A_s= 1.2. The area ratio of the ellipse to the domain size is 0.003. The walls of the cylinder are assumed solid with no-slip boundary conditions. The inlet velocity of the external cross flow is known and the walls of the simulation geometry are assumed to have zero shear stress.

Grid Independency

Since the problem is transient, dynamic mesh is used and the independency of the results from the grid size is examined. For this purpose three different grid sizes are tested for Re=50, laminar flow over stationary cylindrical tube. Lift and drag coefficients are compared and results are shown in Figure 1. As it is observed, all results are acceptable and quite similar, therefore the coarse mesh has been chosen for the simulation with 29056 elements which has 7.6% difference with the fine mesh and is shown in Figure 2.



(a) Drag coefficient



(b) Lift coefficient

Figure 1 Grid independency test.

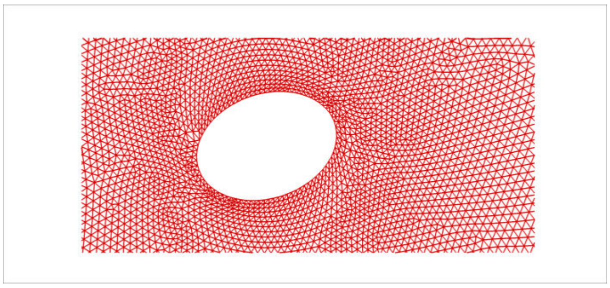


Figure 2 Grid distribution

Validation of Numerical Model

For validating the results, drag coefficient of the mentioned case has been compared with the results of Nakayama [13]. As obtained from Figure 3, for Reynolds number of Re=50, the drag coefficient is equal to 1.605 which confirms the results of the current study with an error smaller than 7.8%.

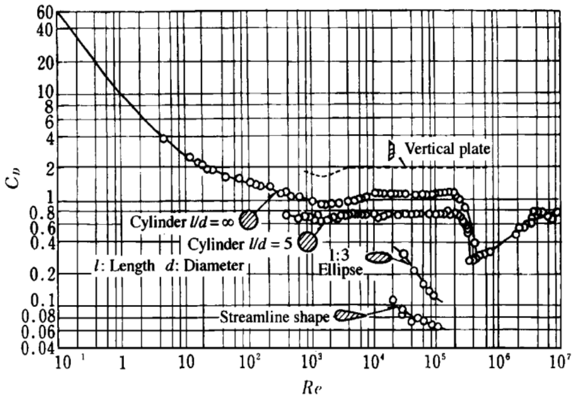


Figure 3 Validating the model by comparing C_D with Nakayama [13].

RESULTS AND DISCUSIONS

Simulations were carried out for two cases: stationary and rotational cylinder, with elliptical cross section. For each case various Reynolds numbers of 10, 25, 50, 100, 200, 300 and 1000 for the external cross flow is considered. Some of the results are shown and discussed in the next sections.

Stationary cylinder:

Pressure field is shown in contour plots of Figure 4 for Re=50, and 1000. The other results are skipped since they are qualitatively similar. As it is expected, a low pressure field is observed downstream and a higher pressure field upstream the flow. Higher the velocity, bigger the pressure difference and for very high Reynolds number pressure variations which cause producing the vortices can be observed from the figure.

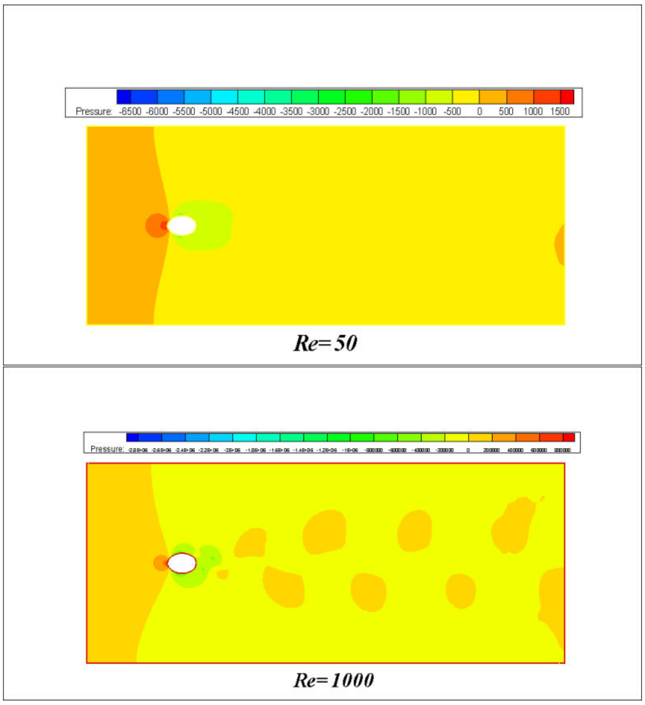


Figure 4 Pressure distribution

Velocity magnitudes and the streamlines are illustrated in Figure 5 for Re=1, 50, 100, and 1000. For very low Reynolds number such as Re=1 to 100, the flow lines have symmetry and the layers of the laminar flow are parallel with no vortices. The hydrodynamic boundary layer formed on the surface of the cylinder is quite visible. The separation, reattachment and the wake regions can be observed in the streamlines for Re=50. When the velocity is increased and the Reynolds reaches to 100 the flow starts oscillating behind the cylindrical body. Though the flow is still laminar, vortices are shaped.

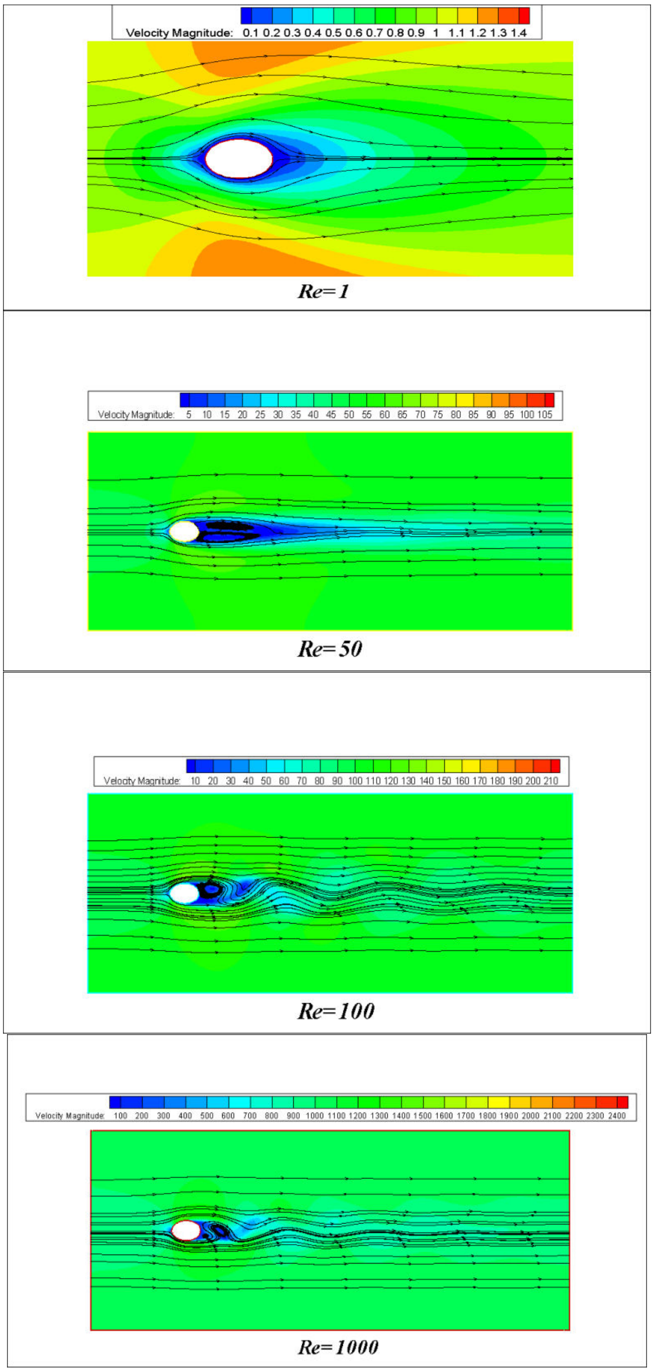


Figure 5 Velocity magnitudes and streamlines for stationary cylinder.

The fluid approaching the cylinder branches out and encircles the cylinder, forming a boundary layer that wraps around the cylinder. The fluid particles on the mid-plane strike the cylinder at the stagnation point, bringing the fluid to a complete stop and thus raising the pressure at that point. The pressure decreases in the flow direction while the fluid velocity increases. At higher velocities, the fluid still hugs the cylinder on the frontal side, but it is too fast to remain attached to the surface as it approaches the top of the cylinder. As a result, the boundary layer detaches from the surface, forming a separation region behind the cylinder. Flow in the wake region is characterized by random vortex formation and pressures much lower than the stagnation point pressure. The high pressure in the vicinity of the stagnation point and the low pressure on the opposite side in the wake produce a net force on the body in the direction of flow. The drag force is primarily due to friction drag at low Reynolds numbers ($Re < 10$) and to pressure drag at high Reynolds numbers ($Re > 5000$). Both effects are significant at intermediate Reynolds numbers. At about $Re < 10$, separation starts occurring on the rear of the body with vortex shedding starting at about $Re < 90$. The region of separation increases with increasing Reynolds number up to about $Re < 10^3$. At this point, the drag is mostly (about 95 percent) due to pressure drag. The drag coefficient continues to decrease with increasing Reynolds number in this range of $10 < Re < 10^3$. [14]

Rotating Cylinder:

For the rotational case, the result of Re=50 are chosen to be presented in this section. The angular velocity is 10 rad/sec counter clockwise, and the dimensionless viscosity of the fluid is unity.

As it is obvious from figure 6, lift and drag coefficients oscillate in time which is due to the elliptical shape of the cross section. If the ellipse diameters get close to each other, the altitude of the oscillations will decrease and they will disappear for the circular cross section (equal radiuses) as it is presented in Figure 1.

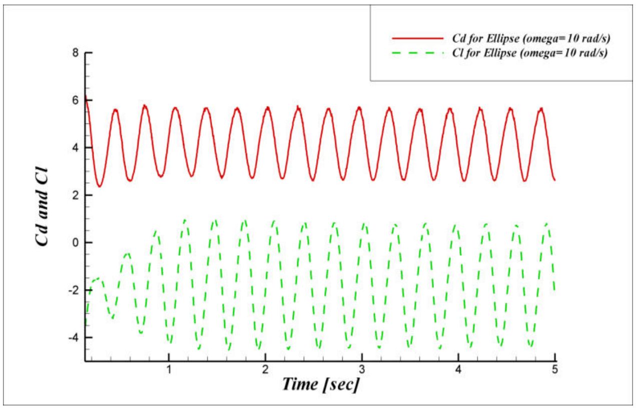


Figure 6 Lift and Drag coefficients, rotating cylinder, Re=50

Pressure contours are illustrated in Figures 7 for Re=50 at time t =0.6 s and Re=1000 at time t=0.5s; and streamlines are presented in Figure 8 for Re=1, 50 and 1000. As it is expected the rotation of the cylinder affect the pressure and the flow filed and the streamlines are deformed compared to the streamlines of the stationary cylinder.

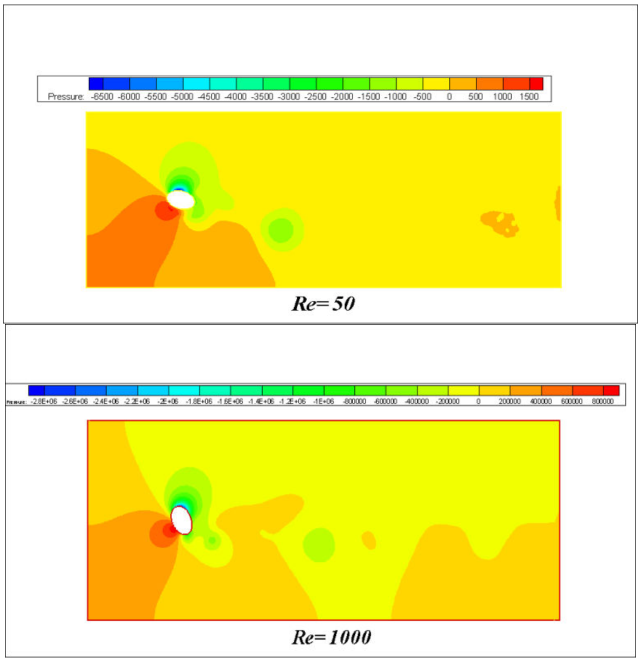


Figure 7 Pressure field for rotating cylinder

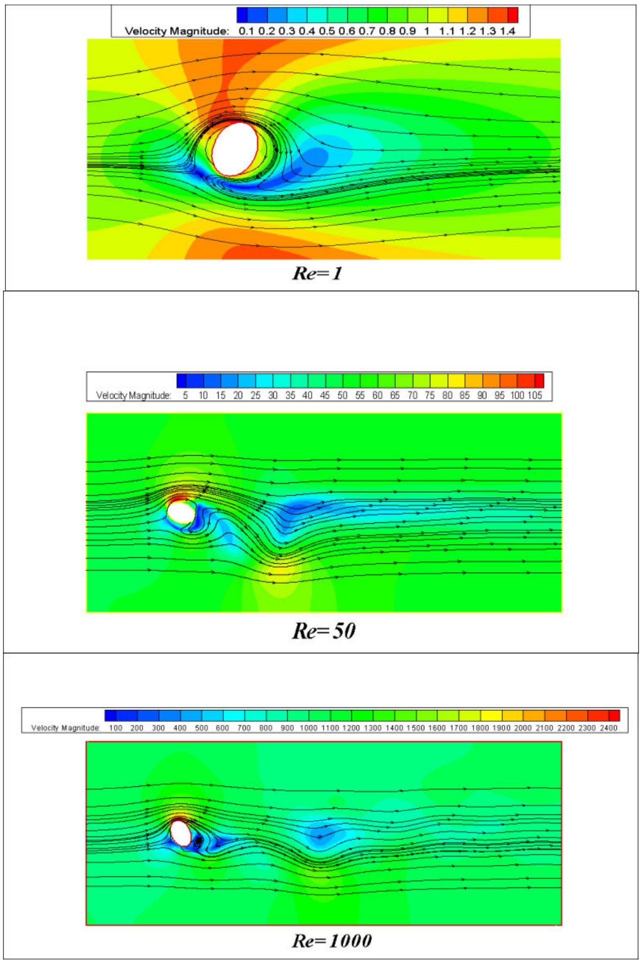


Figure 8 Streamlines for rotating cylinder

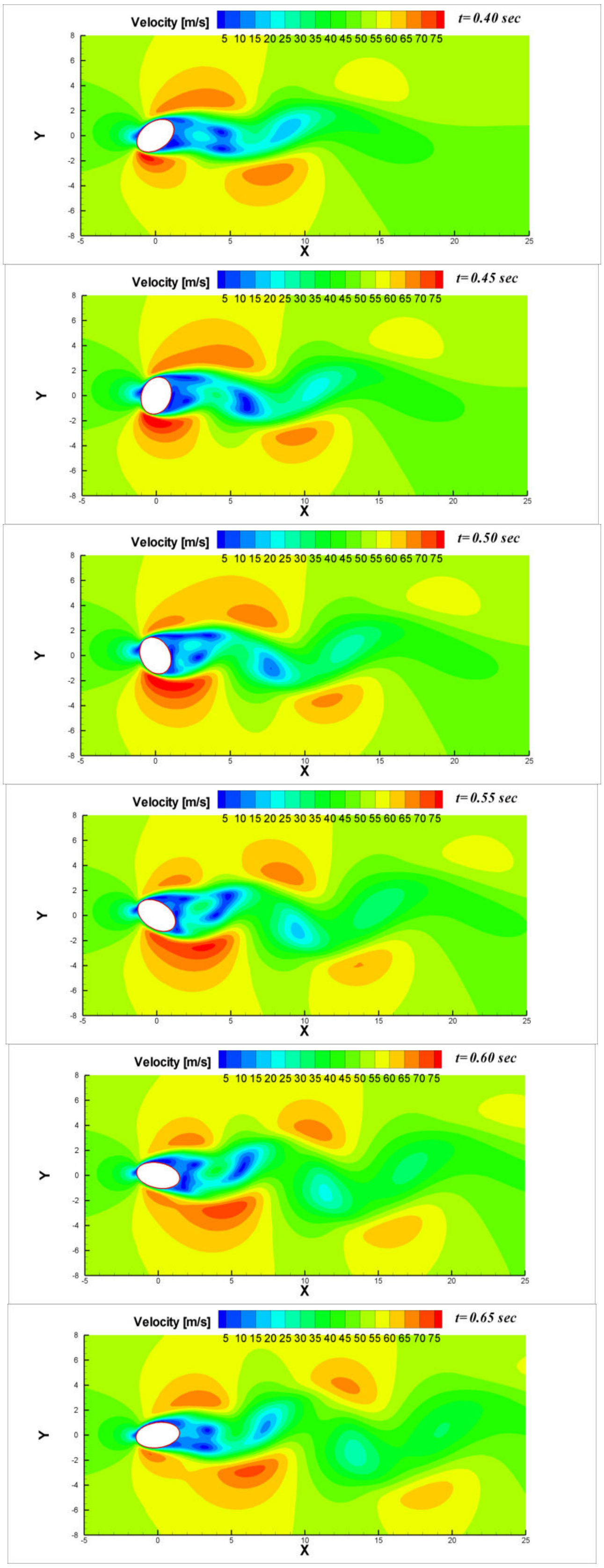


Figure 9 Velocity magnitudes for rotating cylinder

Velocity contours of the rotating cylinder at different times for a complete cycle is shown in Figure 9; the variation of the magnitude of the velocity at the bottom and the top of the cylinder justifies the oscillating behaviour of the lift and drag coefficients and confirms the pressure contour shown in the previous figure.

CONCLUSION

In this study, two-dimensional laminar flow over stationary and rotational cylinder with elliptical cross section is investigated and the lift and drag coefficients as well as pressure and the flow field are studied in detail. It is observed that the lift and drag coefficients perform an oscillatory behaviour when the elliptical-cross-section cylinder is rotating due to the pressure variation in time at the top and below the cylinder.

REFERENCES

[1] Blevins, R.D., Flow Induced Vibration,2nd edition, Van Nostrand Reinhold, 1990

[2] Nobari, M.R.H. and Ghazanfarian J., A numerical investigation of fluid flow over a rotating cylinder with crossflow oscillation, *Int. J. of Computers and Fluids* Vol. 38, 2009, pp. 2026-2036

[3] Schilichting, H., “Boundary Layer Theory”, McGraw-Hill, 1968.

[4] Schindel, L., “Effects of Vortex Separation On Lift Distribution of Elliptic Cylinders”, *J. Aircraft*, Vol. 6, No. 6, 1969, pp 537-543

[5] Lugt, H. J. and Haussling H.J., *J. Fluid Mech*., Vol. 65(4), 1974, pp 771-782

[6] Daube, O. and Ta, P.L., “Etude Numerique D’ecoulements Instationnaires de Fluide Visqueux Incompressible Autourde Corps Profiles Par Une Methode Combinee D’ordere O(h^2) et O(h^4)”, *J. Mec*, Vol. 17, 1979, 651-678

[7] Giorgini, A. and Avci, C., “Impulsively Started Elliptic Cylinder at Re=3000: Catalog of two numerical Experiments”, *Technical Report* CE-HSE-85-11, Hydraulic and systems engineering, Purdue university, West Lafayette, Indiana, 1985.

[8] Hamidi, A. and Giorgini, A., “Numerical Solution of the Navier Stokes Equations: Direct Radial Integration”, *technical report* CE-HSE-85-02, hydraulic and systems engineering, Purdue University, West Lafayette, Indiana, 1985.

[9] Avci, C. and Giorgini, A., “High Order Vortex Formations Past Elliptic Cylinder”, *ASCE Inter divisional Conference*, 1986, 24-44

[10] Mittal, R. and Balachandar, S., “Direct Numerical Simulation of Flow Past Elliptic Cylinders”, *J. of Comput. Physics* Vol. 124, 1996, 351–367

[11] Wang, Z., Liu, J. and Childress, S., “Connection Between Corner Vortices and Shear Layer Instability in Flow Pastan Ellipse”, *Physics of Fluids*, Vol. 11, No. 6, 1999, pp 2446-2448.

[12] Heidarinejad, G. and Delfani, S., “Simulation of Wake Flow Behind an Elliptical Cylinder at High Reynolds Number by Random Vortex Method”, *6th Fluid Dynamic Conference*, Iran University of Science and Technology, Tehran, Iran, 1999, pp 199-210

[13] Nakayama Y. and R. F. Boucher, “Introduction to Fluid Mechanics,” Butterworth-Heinemann, Oxford, 1999

[14] Cengel, Y.A., *Heat Transfer- A Practical Approach*, 5th Ed.