UNSTEADY FLOW AROUND A CIRCULAR CYLINDER: A NUMERICAL SIMULATION

A DISSERTATION

Submitted in partial fulfilment of the requirements for the award of the degree

of

MASTER OF TECHNOLOGY

in

CHEMICAL ENGINEERING

(With Specialization in Computer Aided Process Plant Design)

By

VENKATA NIHARIKA MANGU

DEPARTMENT OF CHEMICAL ENGINEERING
INDIAN INSTITUTE OF TECHNOLOGY ROORKEE
ROORKEE-247 667 (INDIA)

MAY, 2008

CANDIDATE'S DECLARATION

I hereby declare that the work, which is being presented in the dissertation entitled **"UNSTEADY FLOW AROUND A CIRCULAR CYLINDER: A NUMERICAL SIMULATION"** in the partial fulfillment of the requirement for the award of the degree of Master of Technology in Chemical Engineering with specialization in **COMPUTER AIDED PROCESS PLANT DESIGN**, submitted in the Department of Chemical Engineering, Indian Institute of Technology Roorkee, Roorkee, is an authentic record of my own work carried out during the period from July 2007 to May 2008, under supervision of **PROF. SURENDRA KUMAR and DR. AMIT K. DHIMAN**, Department of Chemical Engineering, Indian Institute of Technology Roorkee, Roorkee.

I have not submitted the matter, embodied in this dissertation report for the award of any other degree.

Date: 29 May '08

Place: Roorkee

M.V. Niharika

M.V.NIHARIKA

CERTIFICATE

This is to certify that the above statement made by the candidate is correct to the best of my knowledge and belief

29.5.08

Dr. SURENDRA KUMAR

Professor,

Department of Chemical Engineering

IIT Roorkee

Roorkee, Uttarakhand – 247 667

Dr.AMIT K.DHIMAN

Lecturer,

Department of Chemical Engineering

IIT Roorkee

Roorkee, Uttarakhand – 247 667

ACKNOWLEDGEMENT

I acknowledge the presence of the mighty hand of the LORD in completion of this work successfully.

I am greatly indebted to my supervisors **Dr. Surendra Kumar,** Professor, and **Dr. Amit K. Dhiman,** Lecturer, Department of Chemical Engineering, Indian Institute of Technology Roorkee, for their kind support and guidance during the entire course of this work. Their cooperation and in depth knowledge have made my work possible.

My special thanks are due to **Dr. (Mrs.) Shashi,** Assistant Professor, Department of Chemical Engineering, IIT Roorkee, whose valuable knowledge helped me in understanding the phenomena during this dissertation work.

I would like to thank **Dr. Shri Chand,** Head of the Department of Chemical Engineering, IIT Roorkee for providing various facilities during the course of this work.

I would like to thank Mr.Rahman Jee, Mangeeram Jee, Tripta didi for their kind cooperation. I also thank Akhilesh Jee and the persons of CAD centre for their help by providing computers.

This would be incomplete if I not thank my parents for their consistent support and love.

M.V.Niharika

(M.V.NIHARIKA)

LIST OF FIGURES

LIST OF TABLES

NOMENCLATURE

A_P	Projected area	(m^2)
C_D	Drag coefficient	(dimensionless)
C_{Df}	frictional drag coefficient	(dimensionless)
C_{Dp}	pressure drag coefficient	(dimensionless)
C_L	Lift coefficient	(dimensionless)
C_{Lf}	frictional lift coefficient	(dimensionless)
C_{Lp}	pressure lift coefficient	(dimensionless)
d	diameter of the circular cylinder	(m)
D_∞	Diameter of the outer boundary	(m)
f	Shedding frequency	(1/s)
F_D	Drag force	(N)
F_L	Lift force	(N)
l/D	length to diameter ratio	(m)
p	pressure	(N/m^2)
Re	Reynolds number	(dimensionless)
St	Strouhal number	(dimensionless)
t	time	(s)
r	radius of cylinder	(m)
u, V_x	x direction velocity	(m/s)
v, V_y	y direction velocity	(m/s)
x	Cartesian x coordinate	(m)
X_u, X_d	upstream and downstream distance	(m)

y	Cartesian y coordinate	(m)
V_∞	Uniform velocity at the inlet	(m/s)
σ_x , σ_y	Stress components	(N/m^2)
ρ	density	(kg/m^3)
μ	viscosity	(kg/m.s)
Ψ	stream function	(m^2/s)

INDEX

INTRODUCTION

Flow past bluff bodies is particular matter of concern because of their vast applications in chemical industries and aerodynamics for both steady and unsteady state flows. Typical example for a bluff body that immediately comes to mind is the circular cylinder that has become so popular that number of research papers are devoted to it are far more exceeding than those for all other bluff bodies are featured. Ready availability as piping or tubing, structural convenience and practical importance in engineering has made it a favorable one in laboratory studies. But the primitive reason for its appeal is the simplicity and beauty of the solution. Flow past the circular cylinders in the field of chemical engineering has a wide range of applications such as in heat exchangers. Flow around cylindrical structures is of relevance for many practical applications e.g. offshore risers, bridge piers, periscopes, chimneys, towers, masts, cables, antennae and wires. The aspects of unsteady flows include hydrodynamic propulsion (propeller-hull interaction), flapping wing propulsion in aerodynamics. Recent applications such as the detection of unobserved objects through signature analysis and the migration and dispersion of pollution have intensified the need to understand wake dynamics. The propensity of bluff bodies to vibrate in fluid flow is one of the reasons why this research is important to engineering applications.

1.1 FLOW REGIMES

The Reynolds number plays an important role here because flow separations are often Reynolds number dependent, even if the bodies have sharp edges. The first few regimes designated by Zdrakovich (1997) are creeping laminar state of flow in the range $0 < Re < 4$, laminar flow with steady separation in $4 < Re < 48$ forming a symmetric contra rotating pair of vortices in the near wake, laminar flow with periodic vortex shedding in $48 < Re < 180$. The reason for this dependence is that the state of the boundary layer has a far-reaching influence on the entire (i.e., the global) flow field about a body. In particular, the location of the laminar/turbulent transition point in the boundary layer, or in the separated shear layer, is an important parameter for the phenomena under consideration. Both the state of the boundary layer and the location of transition are responsible for the formation, length, and shape of the separation bubbles, or, more generally, for the topological structure of the flow. When the Reynolds number increases, the following scenario often occurs: a separation bubble is formed; it becomes larger, then unstable and finally then bursts. Changes in the topological structure of the wake can occur in nearly two-dimensional up to three-dimensional flows, even when the body is two-dimensional. The circular cylinder is the classical bluff body that exhibits strong Reynolds number effects. The airfoil mounted at a high angle of attack is interesting because, depending upon the Reynolds number range; it can behave like a bluff body.

Figure 1 illustrates the key role of the laminar/turbulent transition in flow past a circular cylinder, both below and above the critical flow regime. The Reynolds number effects here are clearly evident and dramatic. These phenomena, which are triggered by the transition from laminar to turbulent flow, have characteristics that are universally valid because they occur in similar form in flow over bodies having other cross-sections. To simplify somewhat, we can imagine that the separated shear layer undergoes transition (which occurs in the wake at low Reynolds numbers) at a point that wanders upstream as Reynolds number increases. If, at critical Reynolds numbers, the transition point reaches the surface of the body, separation bubbles could be formed. This process is accompanied by a decrease in the width of the wake (drag crisis) and, because of the unsteady behavior, a jump in the value of the Strouhal number.

For all but the lowest Reynolds numbers the flow past a bluff body results in the asymmetric shedding of vortices into the body's wake. These vortices induce periodic forces on the body and can, if the body is elastically supported, cause it to vibrate. Such a vibration is termed as Vortex-Induced Vibration (VIV). VIV can provide a major source of fatigue damage to engineering structures and it is therefore imperative to predict the occurrence, and the likely amplitude and frequency, of VIV when designing structures that are exposed to current flows.

The flow-induced vibrations of many bluff bodies have been investigated, but by far the most research attention has been given to investigating the VIV of circular cylinders. This is in part due to the engineering importance of circular cross-section bodies. The fluid forces acting on a circular cylinder in both the cross-flow (transverse) direction, the lift, and the stream wise (in-line) direction, the drag, can cause VIV in their respective directions.

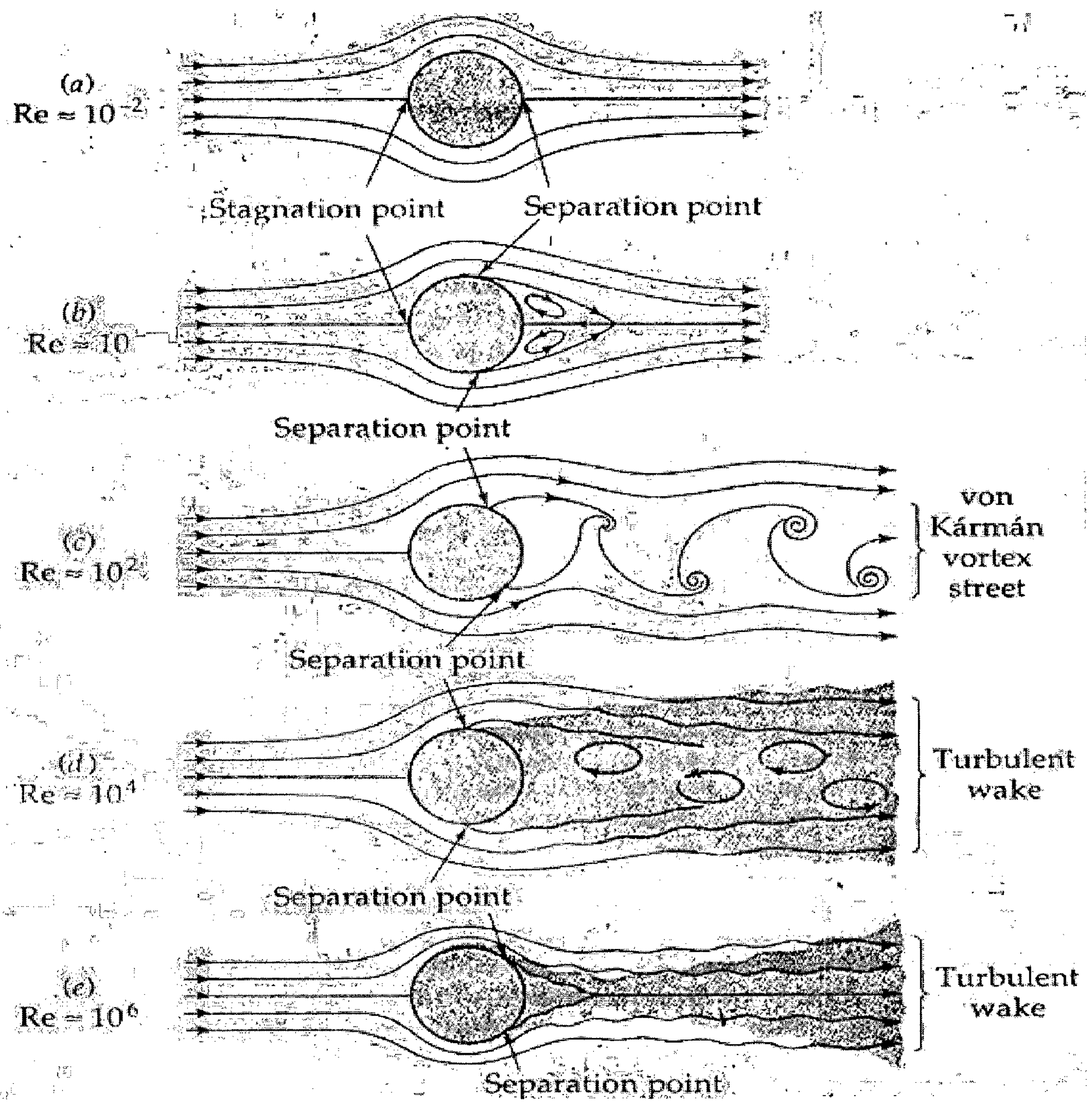

Fig. 1 Flow around circular cylinder at different Reynolds numbers. (Bird et al. 2004)

When the free-stream velocity exceeds a certain critical value in flow past a bluff body, vortex shedding occurs due to the flow instability in the near wake, resulting in periodically oscillating drag and lift forces. Such fluctuating forces may cause structural vibrations, acoustic noise and resonance, which in some cases can trigger structure failure or enhance mixing in the wake. Therefore, it is very important to appropriately control vortex shedding in practical engineering environments.

Uniform flow past a circular cylinder has been accepted as a building-block problem for understanding the vortex dynamics in the wake behind a bluff body and, thus, a considerable number of studies on the uniform flow have been performed so far. In most bluff-body flows of engineering interest, however, the free-stream is not uniform, but sheared. As evidently observed, air and tidal currents have non-zero velocity gradients in space and, thus, can be regarded as sheared. Such examples involve buildings and transport vehicles on the ground and pipelines under the sea.

The aerodynamics of a bluff body in shear flow has been widely investigated in various engineering fields, where the structures are immersed in a planetary boundary layer with a velocity shear. However, in these researches, bluff body models were placed parallel to the plane of velocity shear, which is different from present study. There are many situations in which a bluff body is placed normal to the plane of velocity shear, for instance, bridge decks in an atmospheric boundary layer. The flow around more-or-less bluff bodies is determined by flow separations that form vortex streets in the wake.

1.2 ORGANIZATION OF THESIS

This work is opened with an introduction about the flow over the bluff bodies and its application in industry. Different flow regimes are studied for the cylinder at different ranges of Reynolds numbers in chapter 1. In the chapter 2, a detailed literature is given for the unsteady flow over the cylinder in an unconfined configuration. In chapter 3, a problem statement is formed and the governing equations are presented with boundary conditions along with the solution methodology. In the chapter 4, the results are validated and the drag, lift coefficients and Strouhal number are presented. The conclusions are given in the chapter 5 along with the recommendations.

LITERATURE REVIEW

Many papers have been published on the flow past the circular cylinder for various Reynolds number ranges but this work only includes those studies that are in the Reynolds number range of Re ≤ 500.

Jordan (1972) has studied the oscillatory drag, lift and torque on a circular cylinder in uniform flow for Reynolds numbers 100, 400 and 1000. Numerical computations of the Navier-Stokes equations are caused by the finite difference approximations. He has concluded that the average drag coefficient was larger for the oscillatory wake than the symmetric wake. The relative phase angles for the drag, lift and torque oscillations were given. The period of the drag oscillations was one half of the period of the lift and drag oscillations.

Braza et al. (1986) have studied physical analysis of the pressure and velocity fields in the near wake of a circular cylinder for Reynolds number values of 100, 200, and 1000 numerically. A two dimensional circular cylinder is considered with a domain from its centre at 114.51 of the radius. The numerical simulation is based upon a finite volume velocity pressure formulation of the unsteady Navier Stokes equations. A semi implicit second order accurate scheme is employed. They have concluded that the vortex shedding is an intrinsic phenomenon of the Navier stokes equations and is the response of the dynamic system to the existing perturbations and does not depend on the details of those perturbations. The flow system has then, a strongly deterministic character. The vortex shedding is generated by a less of symmetry of the two dimensional symmetric structures in the wake of the circular cylinder, above a critical value of the Reynolds number owing to physically existing perturbations. As the Reynolds number increases from 200- 1000 there appear secondary vortices in the vicinity of each primary eddy.

Williamson (1988) defined a universal and continuous Strouhal-Reynolds number relationship for the laminar vortex shedding of a circular cylinder for the Reynolds number range of 49 to 178. An experiment was done with cylinders in a 6 in circular test section of an open jet wind tunnel. Oblique – shedding angles were measured by hot wires. He has concluded that if careful attention was paid to damping of any circular cylinder vibrations, and if the stream was sufficiently uniform, the Strouhal curves for parallel shedding should be completely smooth & devoid of discontinuities. If the end conditions are such that the shedding was oblique, then a change on the mode of oblique shedding causes a discontinuity to occur.

Lee and Budwig (1991) have studied the aspect ratio on vortex shedding behind circular cylinder for the range of Reynolds number $50 < Re < 150$. The measurements were performed in a 30x30x183 cm section type low speed tunnel. For cylinders with no end modifications and l/D greater than 60 discontinuity in St-Re curve was found at some point between $64 < Re < 130$. Parallel shedding for cylinder with end pates was generated at $l/D < 43$. The shedding produced in this manner had continuous St-Re curves. They have concluded that the reduction in the l/D ratio stabilizes the wake, i.e., the critical Reynolds number increases and the Strouhal number decreases and also the wake is increased.

Li et al. (1991) have studied laminar flow past one and two circular cylinders for the Reynolds numbers 100 and 200 numerically. In this paper time dependent Navier Stokes equations were solved using the conventional finite element method with velocity pressure formulations. Rectangular domain was considered with 5 as the upstream distance from the centre of the cylinder and 15 as the downstream distance. Two types of boundary conditions are used $u = 1$ and $v = 0$ another type of boundary condition $\sigma_x=0$ and $\sigma_y=0$ was also chosen. It was shown that the first type of boundary condition had a slight influence on drag and lift coefficients. However this influence seemed to be a little greater on the Strouhal number. Nevertheless the two different boundary conditions were found to make no big difference on global flow parameters. They have shown that the drag coefficient was shown to be continuous over time. Its fluctuating component was much smaller than the lift coefficient.

Roshko (1993) have studied the perspectives of bluff body wake dynamics for the Reynolds number range 50 to 180 and above. A model for drag was proposed for bluff plate and theoretically dependence of l/D and C_D on the base suction was determined. He also proposed that with increasing Reynolds number viscous stresses are eventually augmented or replaced by Reynolds stresses which keep the base suction and drag at high levels. But this is too simplistic in estimate because of higher values of Re viscous stresses can be significant in regions of high strain, namely the free shear layers.

Henderson (1995) studied the details of the drag curve near the onset of vortex shedding. The flow field was obtained by solving the two dimensional, time dependent Navier-Stokes equations. These equations were solved approximately by using spectral element method with resolution parameters. Near the onset of vortex shedding the wake exhibits a slow time periodic waviness and the drag changes smoothly. He concluded that the slope of C_D-Re curve changes discontinuously as the flow becomes unsteady.

Fey et al. (1998) have given a new Strouhal- Reynolds number relationship for the circular cylinder in the range 47 < Re < 2 X 10^5. The measurements were carried out in three different facilities, in an Effiel-type wind tunnel with a closed section, in a recirculating water channel and in one meter wind tunnel. For the range of parallel shedding i.e., 47 < Re < 180, a linear relationship was described when Strouhal number was plotted as a function of $1/\sqrt{Re}$ and it was approximated as $St = 0.2684 - \frac{1.0356}{\sqrt{Re}}$. In general, for all the ranges of Reynolds number a relationship of $St = St^* + m/\sqrt{Re}$ was deduced and the constants (St^* & m) were reported. For steady case a two parameter fit and for unsteady case a four parameter fit polynomials were given.

Lange et al. (1998) have studied momentum and heat transfer from cylinders in laminar cross flow at $10^{-4} \leq Re \leq 200$. Finite volume method with a collocated arrangement of the variables was employed for the normalized Navier Stokes equation Pressure equations are solved by SIMPLE algorithm. To improve the accuracy of the numerical results without loss of efficiency local grid refinement is used. Two different types of computational domains and grids were employed: one for lower values of Re and other for larger values. A domain size up to 10^6 times larger than the cylinder diameter was used for the smallest value of Re. For Re > 46 a different type of domain was adopted. A rectangular domain by coupling an O-type grid around the cylinder with an H-type grid in the elongated part of the domain was considered. They concluded that a sudden change in the behavior of the computed values of C_D at Re = 45 when vortex shedding starts and found that they have rightly correlated the Strouhal Re relationship was well fitted with the Williamson (1988) experimental values.

Young et al. (2001) have studied the simulation of laminar vortex shedding flow past cylinders using a coupled boundary element method (BEM) and finite element method (FEM) model for range of Reynolds numbers 50- 140. The model was based on unsteady Navier-Stokes equations in primitive variables solved by the infinite boundary value problems by extracting the boundary effects on a specified finite computational domain using projection method. The external flow field was simulated using the boundary element method. The rectangular domain has 5568 elements and 5750 nodes. At Re = 100 for the fixed cylinder, they found a good agreement with the reported literature.

Norberg (2003) has studied fluctuating lift on a circular cylinder for Reynolds number range 47 to 2 x 10^5. On the basis of this review article, he summarized that both Strouhal number and rms lift coefficient increases rapidly within the laminar shedding. At the highest attainable Reynolds number (i.e., Re=190) for 2D flow, the rms lift coefficient and Strouhal numbers are 0.45 and 0.19 respectively.

Ayyalasomayajula et al. (2003) have studied analysis of higher-order compact differencing scheme by studying flow past a circular cylinder in the Reynolds number ranging from 100 to 1000. The 3D unsteady incompressible Navier Stokes equation and used flow solver FDL3DI is used to investigate the separated flow. They have shown that their scheme of solution is in good accordance with the past results.

Baranyi (2003) did computation of unsteady momentum and heat transfer for a fixed circular cylinder in laminar flow for the Reynolds number range from 50 to 180. Navier Stokes and Poisson's equations were used and finite difference method was used to solve the above equations. The diameter of the outer boundary of computation is 30 times larger than the diameter of the cylinder and O type grid is used. Drag and lift coefficients and also Strouhal number variations with the Reynolds number are computed and found that they had a good agreement with the experimental results even the 3D wake formation at about Re = 160.

The momentum transfer characteristics of the power-law fluid flow past an unconfined elliptic cylinder was investigated by **Sivakumar et al. (2006).** The numerical study was carried out by solving the Navier-Stokes equations using FLUENT software. A grid size of 24000 grid points is considered. They computations carried out here are for the Reynolds number of range $0.01 \leq Re \leq 40$ and for different aspect ratios of 0.2, 0.5, 1, 2 and 5. They have concluded that the total drag coefficient on the power-law index was seen stronger in shear thinning fluids than that in shear thickening fluids. They have also concluded that the wake tends to be smaller in shear thickening fluids for aspect ratio less than 1 than that in shear thickening fluids for aspect ratio greater than 1. At low Reynolds numbers, the front stagnation pressure coefficient was always higher in shear-thinning fluids than that in shear-thickening fluids.

A systematic approach to the numerical calculation of fundamental quantities of the two dimensional flow over a circular cylinder was studied by **Posdziech and Grudman (2007).** Two different rectangular domain sizes were considered for steady and unsteady cases. Numerical simulations of the flow around the cylinder at Re = 5 to 120 were carried out using spectral element method. A mesh of 186 spectral elements was taken for l/D ratio 20 to 120 and 246, 306, and 396 were taken for l/D = 500, 2000 and 4000 respectively. A polynomial expression of the drag and base pressure coefficients was given which can be used to estimate errors caused by blockage effects due to finite domain extensions. They have shown that the unsteady drag variation was smaller compared to the steady case and the lift coefficient strongly increased with Reynolds number. The Reynolds number dependency on drag, lift and base pressure coefficient's and Strouhal-Reynolds relationship were given polynomial approximations up to fifth order.

A numerical simulation of the flow past a circular cylinder which was able to oscillate transversely to the incident stream at the Reynolds number 100 was presented by **Placzek et al. (2008).** The 2D Navier Stokes equations are solved by finite volume method with an industrial CFD code in which a coupling procedure has been implemented in order to obtain the cylinder displacement. In this paper a preliminary work was first conducted for a fixed cylinder to check the wake characteristics for Reynolds number smaller than 150 in the laminar regime. The periodicity of the shedding leads naturally to the fluctuation of the aerodynamics coefficients. The periodic state reached was characterized by the oscillations of the drag coefficient at the twice the lift frequency. The fluctuations of the suction coefficient are governed by two frequencies: the main frequency was equal to the lift oscillation frequency and the secondary was identical to the drag oscillation frequency.

Numerical simulation of laminar flow past a circular cylinder was studied by **Rajani et al. (2008).** In this simulation, an implicit pressure-based finite volume method was used for time-accurate computation of incompressible flow using second order accurate convective flux discretization schemes. The calculations use 144 nodes stretched along the radial direction and 121 equispaced nodes around the circumferential direction for 2D and for the case of 3D, 32 equidistant parallel planes cover the total length of 2πD and periodic boundary conditions are used at the end planes. The critical Reynolds number at which the cylinder wake with a symmetric vortex pair becomes unstable was determined and it is reported to be 47.4. The temporal evolution of the lift and drag coefficients computed separately from both two and three dimensional simulation of the flow at five Reynolds numbers from 100 in step of 100 till 400. They investigated the initial transients die down in the first few steps and the flow eventually reaches a periodic but statistically stationary state. Up to Re = 200, the two and three dimensional computation results are observed to be almost overlapping, showing no significant difference in the temporal variation of the lift and drag coefficients at the stationary state.

Based on the above literature, a large number of studies are available for the unsteady flow across cylinder of circular cross section. However, only a few results have been reported for the unsteady periodic flow regime. Therefore, the main objective of this work is to investigate the unsteady (periodic) flow regime for the range $60 \leq Re \leq 150$. The other objective is to provide the data base for drag & lift coefficients and Strouhal number for the above range of conditions. In addition, flow patterns have been presented by stream line profiles.

PROBLEM STATEMENT & MATHEMATICAL FORMULATION

3.1 PROBLEM STATEMENT

Consider the two dimensional, unsteady flow of an incompressible fluid with a uniform velocity V_∞ across a circular cylinder. The unconfined flow condition is simulated here by creating an artificial circular boundary of diameter D_∞, as shown in Fig. 2. The outer boundary is taken to be sufficiently large such that the end effects are negligible.

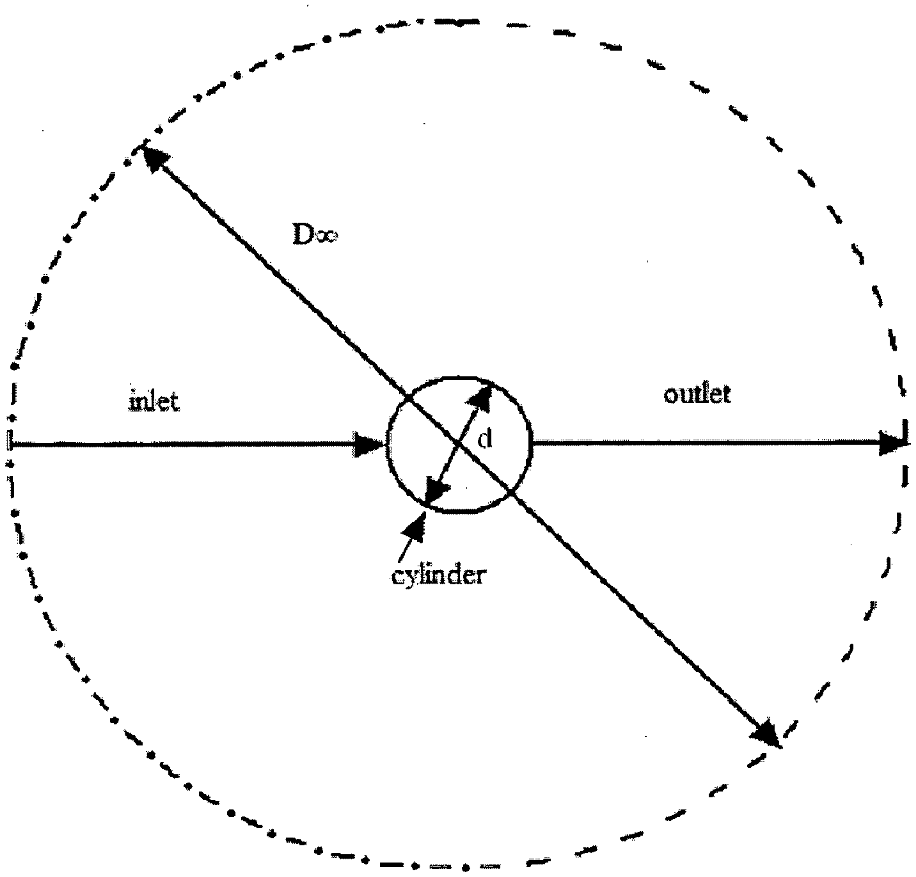

Fig. 2 Schematic of flow around the circular cylinder

3.2 GOVERNING EQUATIONS

The flow around the cylinder is simulated by solving continuity and x- & y- components of Navier-Stokes equations:

Continuity equation:

$$\frac{\partial V_x}{\partial x} + \frac{\partial V_y}{\partial y} = 0. \tag{1}$$

X – momentum equation:

$$\frac{\partial V_x}{\partial t} + \frac{\partial V_x V_x}{\partial x} + \frac{\partial V_y V_x}{\partial y} = -\frac{\partial p}{\partial x} + \frac{1}{Re}\left(\frac{\partial^2 V_x}{\partial x^2} + \frac{\partial^2 V_x}{\partial y^2}\right) \tag{2}$$

Y – momentum equation:

$$\frac{\partial V_y}{\partial t} + \frac{\partial V_y V_x}{\partial x} + \frac{\partial V_y V_y}{\partial y} = -\frac{\partial p}{\partial y} + \frac{1}{Re}\left(\frac{\partial^2 V_y}{\partial x^2} + \frac{\partial^2 V_y}{\partial y^2}\right) \tag{3}$$

Where Re is the Reynolds number defined as Re = $\frac{\rho d V_\infty}{\mu}$

3.2 BOUNDARY CONDITIONS

The physically realistic boundary conditions for this flow may be written as follows:

- *At the inlet boundary*:

The condition of uniform flow is imposed, i.e.,

$$V_x = V_\infty \text{ and } V_y = 0. \tag{4a}$$

- *On the surface of the circular cylinder*:

The standard no-slip condition is used, i.e.,

$$V_x = 0 \text{ and } V_y = 0. \tag{4b}$$

- *At the exit boundary*:

The default outflow boundary condition in FLUENT, which assumes a zero diffusion flux for all flow variables, was used. Physically this condition implies that the conditions of the outflow plane are extrapolated from within the domain and have negligible impact on the upstream flow conditions. This is similar to Neumann boundary condition.

$$\frac{\partial V_x}{\partial x} = 0, \ \frac{\partial V_y}{\partial x} = 0. \tag{4c}$$

These equations (1-3) along with the boundary conditions [equations (4a-4c)] have been used to solve the flow around the cylinder and the flow field has been represented by streamline profiles. The parameters such as C_D, C_L, St etc. have been calculated as follows:

The conventional drag coefficient $C_D \left(= \frac{F_D/A_p}{1/2 \rho V_\infty^2}\right)$ has two parts, one is the frictional drag and the other one is the pressure drag. Similarly the lift coefficient $C_L \left(= \frac{F_L/A_p}{1/2 \rho V_\infty^2}\right)$ has frictional lift and pressure lift.

$$C_D = C_{Df} + C_{Dp} \tag{5}$$

$$C_L = C_{Lf} + C_{Lp} \tag{6}$$

In the unsteady flow regime, the most frequently used flow quantity is the non-dimensional frequency of vortex shedding as defined by

$$St = \frac{fd}{V_\infty} \tag{7}$$

Where 'St' is the Strouhal number and 'f' is the vortex shedding frequency.

Stream function is calculated by $V_x = -\frac{\partial \Psi}{\partial y}$ and $V_y = \frac{\partial \Psi}{\partial x}$ (8)

3.4 GRID STRUCTURE AND DOMAIN STUDY

In the present study, the grid structure has been created in GAMBIT. It is the preprocessor to FLUENT. The grid here consists of non-uniform cells having a total of 24000 grid points in the computational domain. The grid near the surface of the cylinder is sufficiently fine to resolve the boundary layer around the cylinder. The nearest grid point from the object is at a distance of 0.0015d. These values have been chosen based on the results reported elsewhere (Sivakumar et al., 2006).

3.5 METHOD OF SOLUTION

In the present study, numerical computations have been carried out using finite volume method (FVM) based commercial CFD package FLUENT. The structured quadrilateral cells of non-uniform spacing were generated using GAMBIT. Furthermore, the two dimensional unsteady solver was used with second order upwinding scheme for the convective terms in the momentum equations. However, the diffusive terms have been discretized using the central difference scheme. The semi implicit method for the pressure linked equations (SIMPLE) has been used here. A convergence criterion of 10^{-15} was used for continuity, *x*- and *y*- component equations.

RESULTS AND DISCUSSION

In this work, unsteady flow computations have been carried out by varying the Reynolds numbers of 60, 100 and 150. The results here are based on the domain size of 300d and the total number of grid points of 24000. Based on the literature (Williamson, 1988 and Zdrakovich, 1997), the flow for the above range of conditions would be unsteady periodic in nature.

4.1 VALIDATION OF RESULTS

The following Tables (1-3) show the validation of the present results with the literature values.

Table 4.1: Validation of C_D and St for Re = 60

Author	C_D	**St**
Baranyi (2003)	1.42	0.14
Posdziech and Grundmann (2007)	1.36	0.134
Placzek et al. (2008)	1.45	0.14
Present work	**1.36**	**0.133**

In the above table (Table 1), we can see that the present results are in excellent agreement with Posdziech and Grundmann (2007), but there is a slight difference in the value of the drag coefficient with the results of Baranyi (2003) and Placzek et al. (2008). The possible reason may be explained as the domain that was considered by Baranyi (2003) was 30d and a rectangular domain was considered with an upstream distance of 12.5d and the downstream of 20d by Placzek et al. (2008) as opposed to the domain of 300d in the present study. Therefore, the end effects are more pronounced in their studies than that of the present work.

Table 2: Validation of C_D and St for Re = 100

Author	C_D	**St**
Rajani et al. (2008)	1.34	0.16
Posdziech and Grundmann (2007)	1.31	0.163
Baranyi (2003)	1.35	0.163
Ayyalasomayajula et al. (2003)	1.33	0.164
Lange et al. (1998)	-	0.165
Li et al. (1990)	1.33	0.163
Braza et al. (1986)	1.28	0.16
Present work	**1.30**	**0.162**

The comparison of C_D and St at Re = 100 can be seen in Table 2. It can be seen that the Strouhal number is in excellent agreement with all the studies and the slight difference in the value of drag coefficient is due to the domain and grid sizes (see Table 3).

Table 3: Domain and Grid sizes of different studies

Author	Domain	Grid size
Rajani et al. (2008)	40d	124x144
Braza et al. (1986)	114.55r	13530
Li et al. (1990)	$X_u = 5d$, $X_d = 15d$	-
Lange et al. (1998)	$X_u = 30d$, $X_d = 100d$	-
Ayyalasomayajula et al. (2003)	-	155x101, 155x201
Present work	**300d**	**24000**

Table 4: Validation of C_D and St for Re = 150

Author	C_D	**St**
Posdziech and Grundmann (2007)	1.27	0.183
Baranyi (2003)	1.33	0.185
Present work	**1.29**	**0.181**

Similarly comparison of C_D and St values at Re = 150 can be seen in Table 4. Once again an excellent agreement can be seen with Posdziech and Grundmann (2007). However, a slight difference in the value of drag coefficient due to the earlier mentioned reason (Table 4.1) has been observed with Baranyi (2003).

4.2 FLOW PATTERN

4.2.1 Instantaneous streamline profiles

Figure 3 shows the instantaneous streamline profiles for one complete cycle where the cycle time is divided into 6 equal parts. At each time interval, the streamline contours are plotted. The figure clearly shows that vortices are detached from the surface of the cylinder alternatively. At $t = t_0$ a wake is generated and then at time at $t = t_1$ it grows into a bigger size and still more increases in size and another wake is seen generated at the upper side of the cylinder. Further, it grows and finally the profile is same at the end as at $t = t_0$.

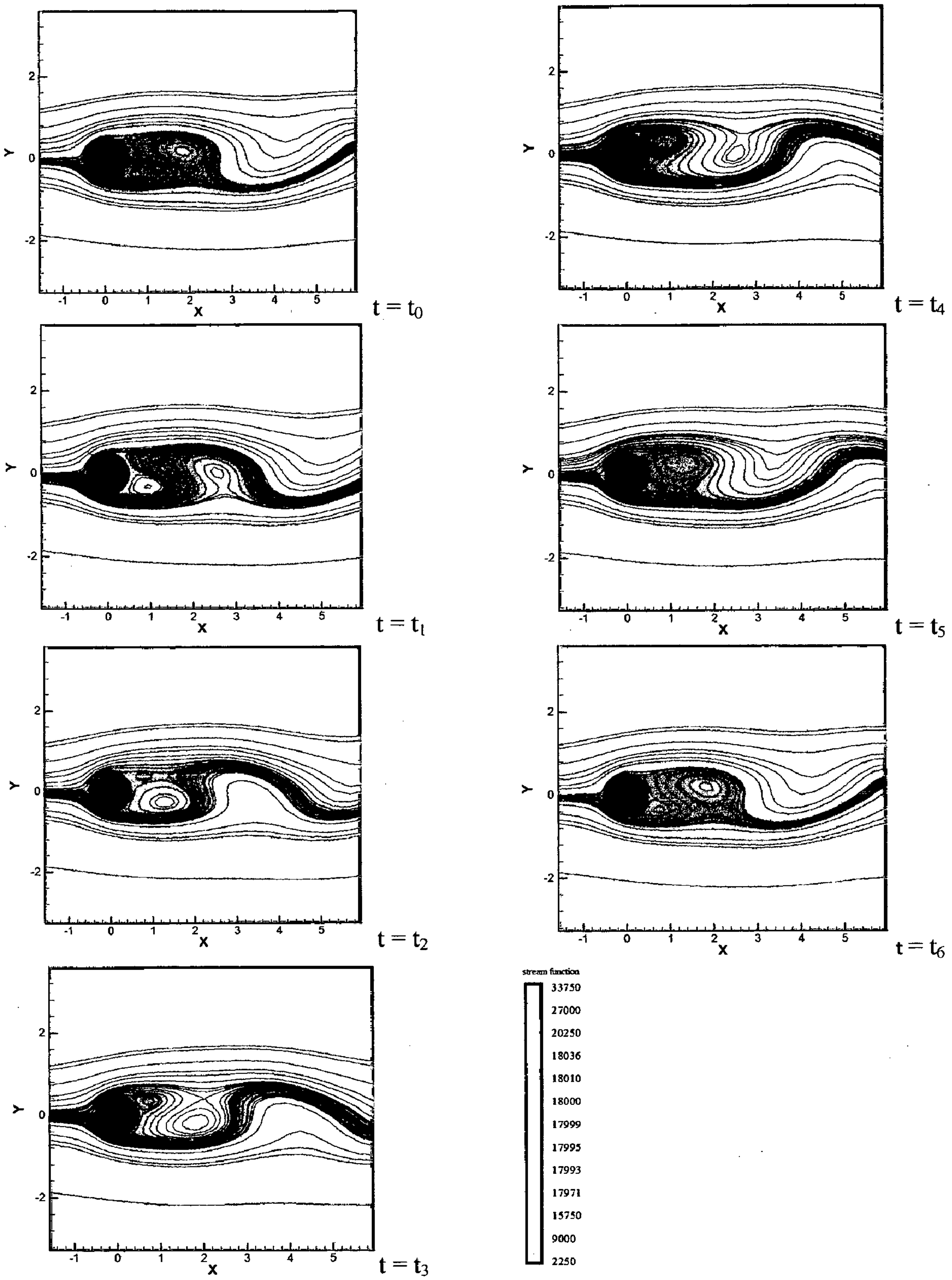

Fig. 3 Instantaneous stream line profiles at Re = 60.

4.3 DRAG COEFFICIENT

The drag coefficient for the Reynolds numbers 60, 100 and 150 are generated with the time (Figs. 4-6). The below figures are the time histories of the drag coefficient. As the time increases the drag starts to oscillate in a fixed period for all the cases.

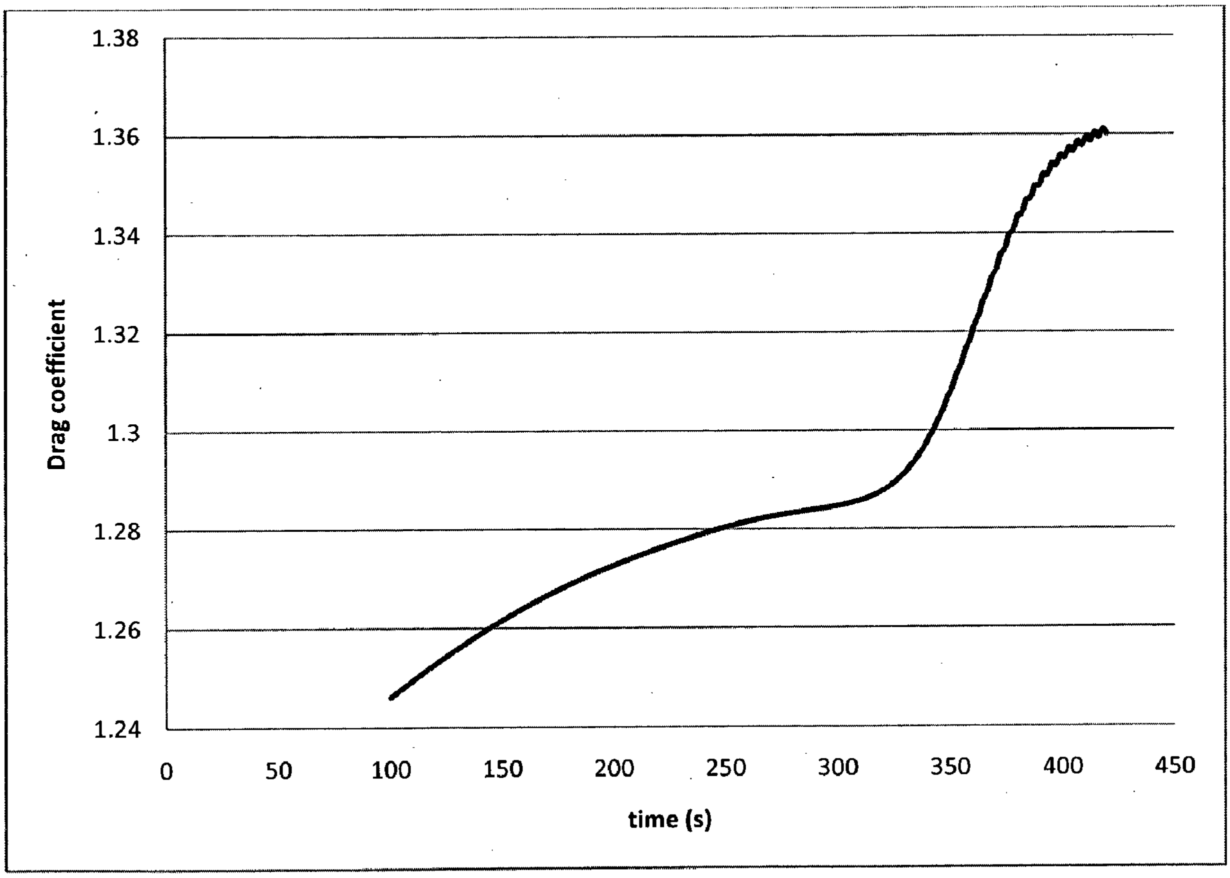

Fig. 4 Time history of drag coefficient at Re = 60

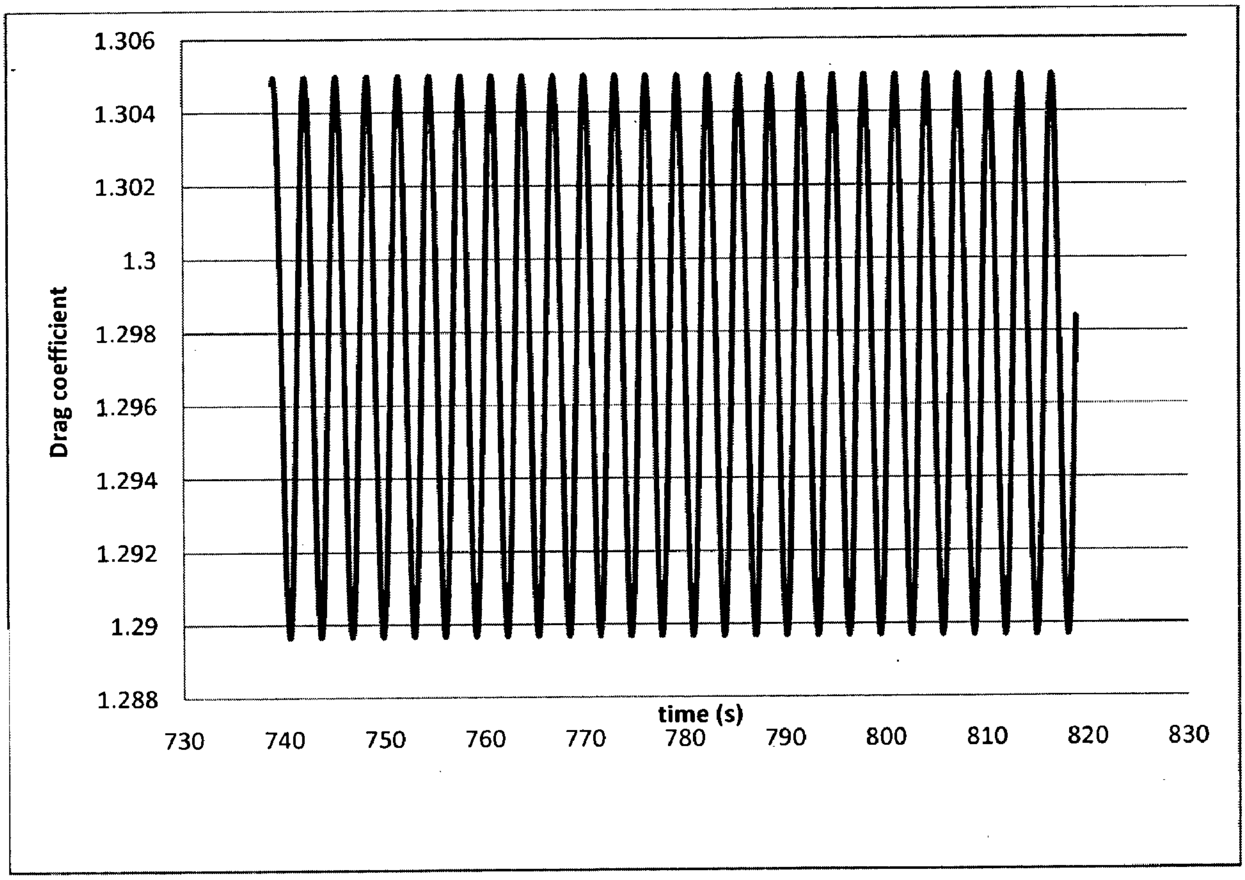

Fig. 5 Time history of drag coefficient at Re = 100

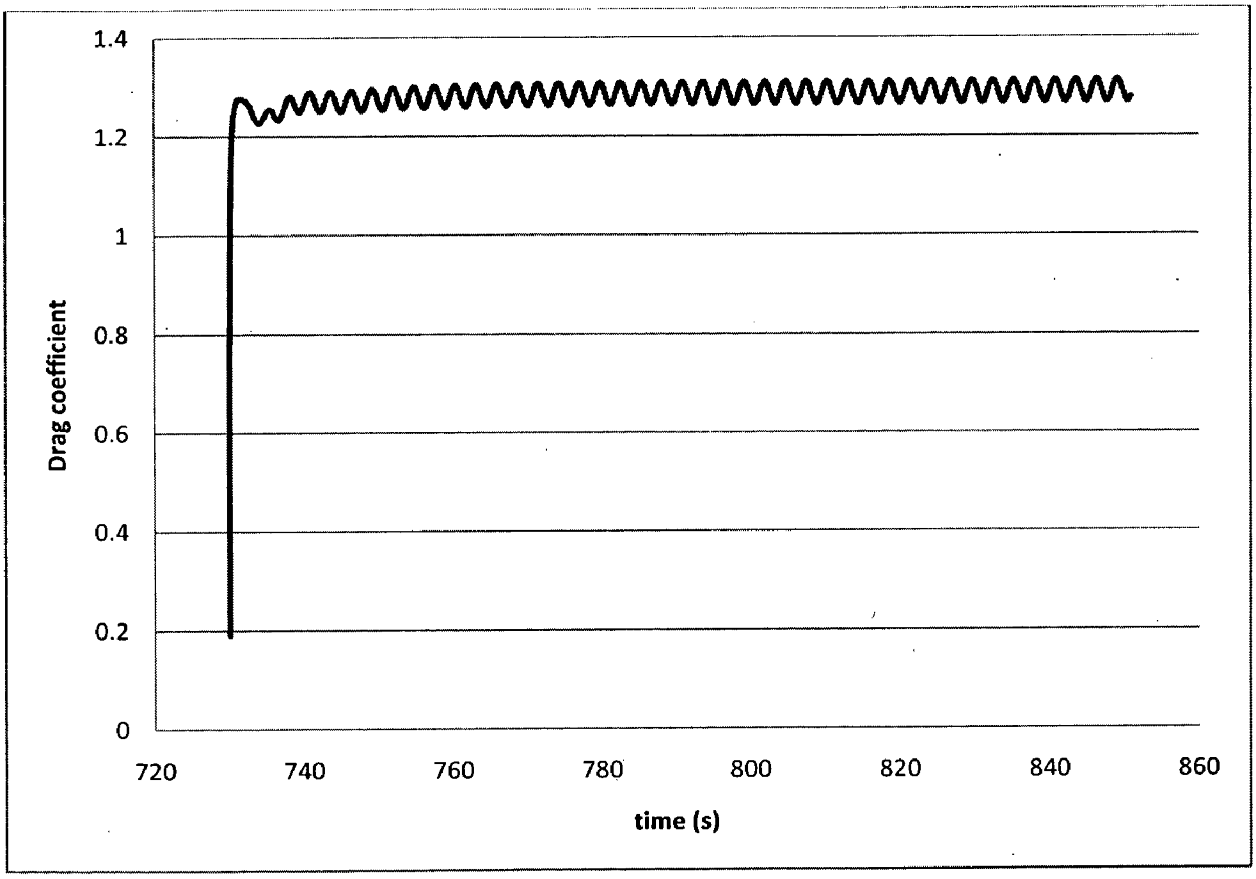

Fig. 6 Time history of drag coefficient at Re = 150

4.4 LIFT COEFFICIENT

As in the case of drag coefficient, the lift coefficient also oscillates with more amplitude than the drag (Figs. 7-9). The following figures show the time histories of the lift coefficients for the above range of Reynolds numbers.

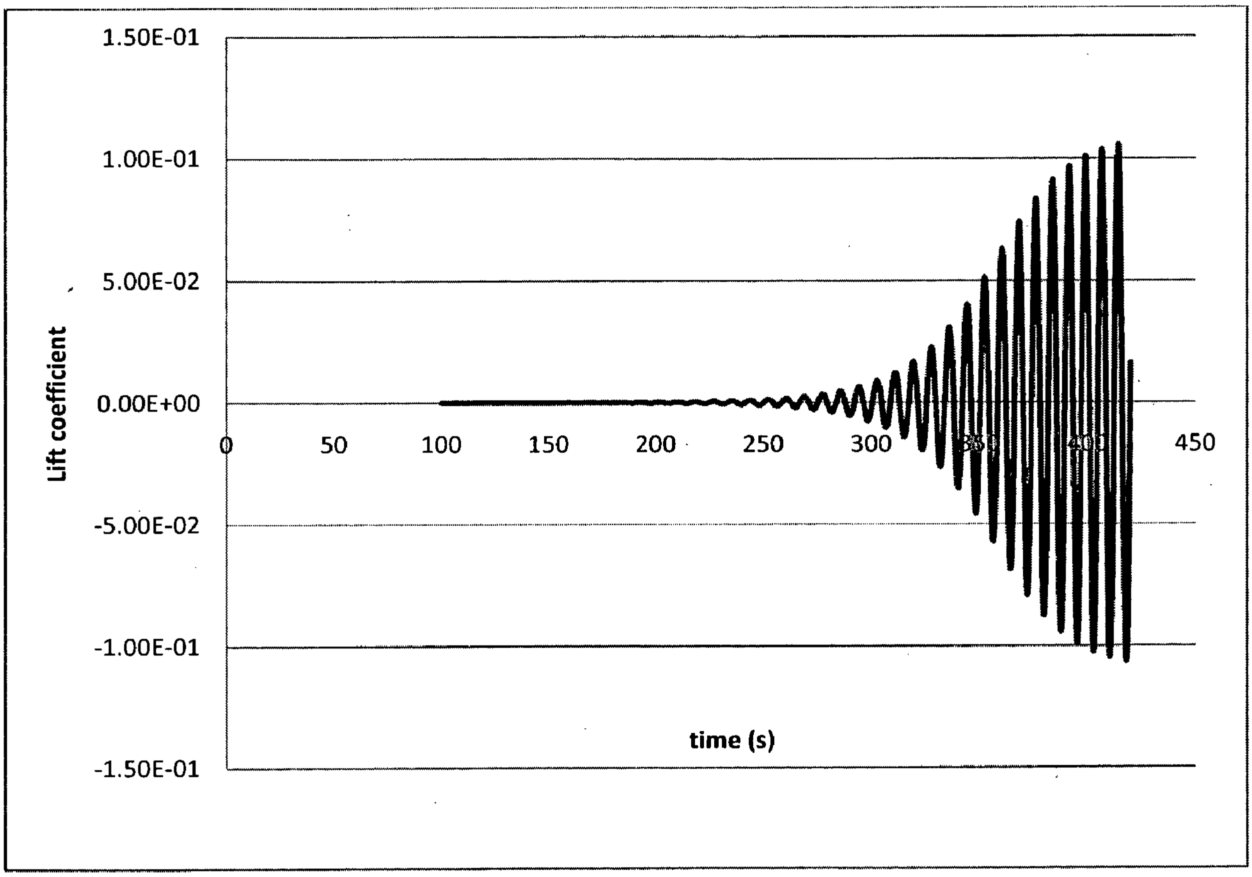

Fig. 7 Time history of lift coefficient at Re = 60

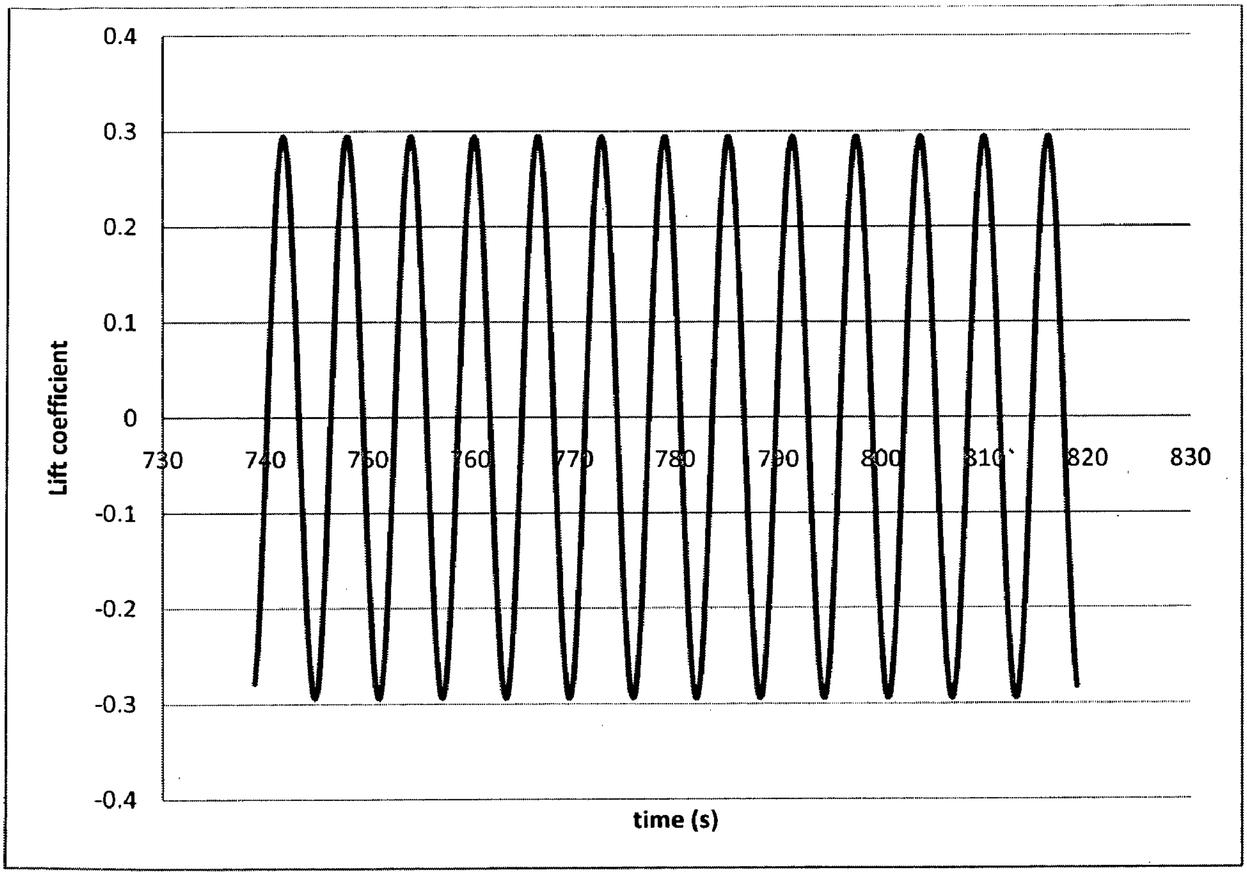

Fig. 8 Time history of lift coefficient at Re = 100

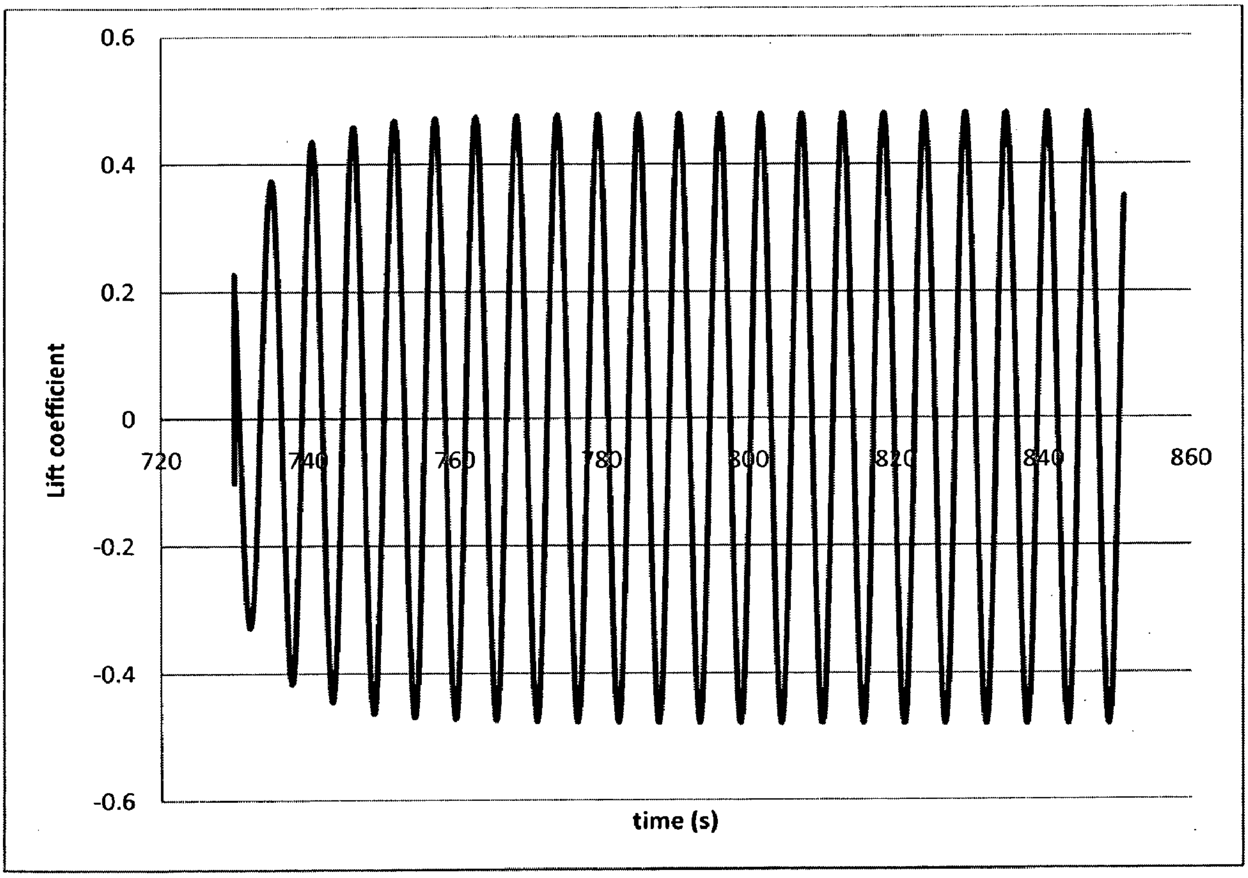

Fig. 9 Time history of lift coefficient at Re = 150

CONCLUSIONS AND RECOMMENDATIONS

5.1 CONCLUSIONS

In the present study, 2D flow of water like fluid around a cylinder has been simulated by using commercial software package FLUENT. The finite volume method has been implemented to solve the governing equations. For the descretization of convective terms, a second order upwinding scheme is used and the central difference scheme has been used for diffusive terms. The orthogonal structured grid has been created by using GAMBIT. The validation of the present results for C_D and St has also been presented. Time history profiles of drag and lift coefficients are presented for the Reynolds numbers of 60, 100 and 150. The flow field is characterized by instantaneous streamline profiles. As the Reynolds number increases, the drag coefficient decreases and the shedding frequency increases for the range of conditions covered here.

5.2 RECOMMENDATIONS FOR FUTURE WORK

1. In this work, unconfined problem is considered and it can be extended to confined flow configuration.
2. This study deals only with the momentum transport. Thus, it can also be extended to energy study.

REFERENCES

Ayyalasomayajula,H., Mutnuri, P.K., Ghia, K., Ghia, U., 2003. Analysis of higher-order differencing scheme by studying flow past a circular cylinder. Second MIT Conference on Computational Fluid and Solid Mechanics 830-833.

Baranyi, L., 2003. Computation of unsteady momentum and heat transfer from a fixed circular cylinder in laminar flow. Journal of Computational and Applied Mechanics 4, 13-25.

Bird, R.B., Stewart, W.E., Lightfoot, E.N., 2004 Transport Phenomena. Wiley Student Edition. Second Edition.

Braza, M., Chassaing, P., Ha minh, H., 1986. Numerical study and physical analysis of the pressure and velocity fields in the near wake of a circular cylinder. Journal of Fluid Mechanics 165, 79-130.

Fey, U., Koing, M., Eckelmann, H., 1998. A new Strouhal-Reynolds-number relationship for the circular cylinder in the range $47 < Re < 2 X 10^5$. Physics of Fluids 10, 1547-1549.

Henderson, R.D., 1995. Details of the drag curve near the onset of vortex shedding. Physics of Fluids 7, 2102-2104.

Jordan, S.K., 1972. Oscillatory drag, lift and torque on a circular cylinder in a uniform flow. Physics of Fluids 15, 371-376.

Lange, C.F., Durst, F., Breuer, M., 1998. Momentum and heat transfer from cylinders in laminar cross flow at $10^{-4} \leq Re \leq 200$. International Journal of Heat and Mass Transfer 41, 3409-3430.

Lee, T., Budwig, R., 1991. A study of aspect ratio on vortex shedding behind circular cylindes. Physics of Fluids 3, 310-315.

Li, J., Chambarel, A., Donneaud, M., Martin, R., 1991. Numerical study of laminar flow past one and two cylinders. Computers and Fluids 19, 155-170.

Norberg, C., 2003. Fluctuating lift on circular cylinder: review and new measurements. Journal of Fluids and Structures 17, 57-96.

Placzek, A et al., Numerical solution of an oscillating cylinder in a cross flow at low Reynolds number: Forced and free oscillations, Computers and Fluids (2008), doi:10.1016/j.compfluid.2008.01.007

Posdziech, O., Grundmann, R., 2007. A systematic approach to the numerical calculation of fundamental quantities of the two-dimensional flow over a circular cylinder. Journal of Fluids and Structures 23, 479-499.

Rajani, B.N., et al., Numerical simulation of laminar flow past a circular cylinder, Applied Mathematical Modelling. (2008), doi:10.1016/j.apm.2008.01.017

Roshko, A., 1993. Perspectives on bluff body aerodynamics. Journal of Wind Engineering and Industrial Aerodynamics 49, 79-100.

Sivakumar, P., Bharti, R.P., Chhabra, R.P., 2006. Steady flow of power-law fluids across an unconfined elliptical cylinder. Chemical Engineering Science 62, 1682-1702.

Williamson, C.H.K., 1988. Defining a universal and continuous Strouhal-Reynolds number relationship for the laminar vortex shedding of a circular cylinder. Physics of Fluids 10, 2742-2744.

Young, D. L., Huang, J.L., Eldho, T.I., (2001) Simulation of laminar vortex shedding flow past cylinders using a coupled BEM and FEM model. Computer Methods in Applied Mechanics and Engineering 190, 5975-5998.

Zdrakovich, M.M., 1997. Flow around Circular Cylinders. Fundamentals, Vol. 1. Oxford University Press, New York, USA.