Numerical Simulation of Unconfined Flow past a Square Cylinder with Forced Convection Heat Transfer

[1]**Mr. Avinash Dholiwal**
[1]Assistant Professor
[1]Mechanical Engineering Department,
[1]Amity University Gurgaon, Haryana, India

Abstract : ***Flow past a square cylinder has been studied extensively for over a century, because of its interesting flow features and practical applications. This problem is of fundamental interest as well as important in many engineering applications. The characteristics of flow around a square cylinder placed at symmetric condition are governed by the Reynolds number (Re). In the present study two dimensional simulations of flow past a square cylinder have been carried out for a Reynolds number of up to 160. The modeling of the problem is done by GAMBIT 2.3 preprocessing software. The computations are carried out using a commercial CFD solver, FLUENT 6 .3, which uses a finite volume approach to discretise governing and model equations for incompressible laminar flow. The average axial and transverse velocities downstream of the cylinder show good matching with the experimental results. The Recirculation Length, velocity profile, Isotherms pattern and Velocity contours have been plotted and compared with previous studies available in literature. The result shows reasonably good matching.***

IndexTerms – Heat Transfer, Convection, Reynolds's number.

1. Introduction

The phenomenon of flow separation and bluff body wakes has long been intensively studied because of its fundamental significance in flow physics and its practical importance in aerodynamic and hydrodynamic applications. The flow of fluid past cylinders of various cross sections represents an idealization of several industrially important applications. It is readily acknowledged that a systematic study of the flow past a single cylinder not only provides valuable insights into the nature of flow, but also serves as a useful starting point to understand the flow in real-life multi-cylinder and other applications such as flow past pipelines near the ground, flow past building construction, suspension bridge, heat transfer enhancement in heat exchangers and forced-air cooling of board-mounted electronic components etc.

The Reynolds's number represents the ratio of inertial forces to viscous forces of flow thus consequently quantifies the relative importance of these two types of forces for given flow conditions. $Re = \frac{\rho VB}{\mu}$ Where V is the air velocity and B is the obstacle width, Reynolds number is used to characterize different flow regimes, such as laminar or turbulent flow At large Reynolds numbers, the inertia forces, which are proportional to the density and the velocity of the fluid, are large relative to the viscous forces, and thus the viscous forces cannot prevent the random and rapid fluctuations of the fluid.

1.1 Computational Fluid Dynamics (CFD)

CFD is the systematic application of computing systems and computational solution techniques to mathematical models formulated to describe and simulate fluid dynamic phenomena.CFD is part of computational mechanics, which in turn is part of simulation techniques. Simulation is used by engineers and physicists to forecast or reconstruct the behavior of an engineering product or physical situation under assumed or measured boundary conditions (geometry, initial states, loads, etc.). A variety of reasons can be cited for the increased importance simulation techniques have achieved in recent years:

- Need to forecast performance
- Cost and/or impossibility of experiments
- The desire for increased insight
- Advances in computer speed and memory

Advances in solution algorithms

2 Problem Formulation and Turbulence Modeling

2.1 Statement of Problem:- In the present problem 2-D simulations of the unconfined flow past a square cylinder with forced convection heat transfer have been carried out up to Reynolds number 160 for different cylinder widths (B =1, 2 & 3). The dimensions of the geometry are

B = width of square cylinder
L = length of domain
L_a = distance between the inlet and front surface of square cylinder
L_t = distance between the exit and rear surface of square cylinder
H = height of the domain

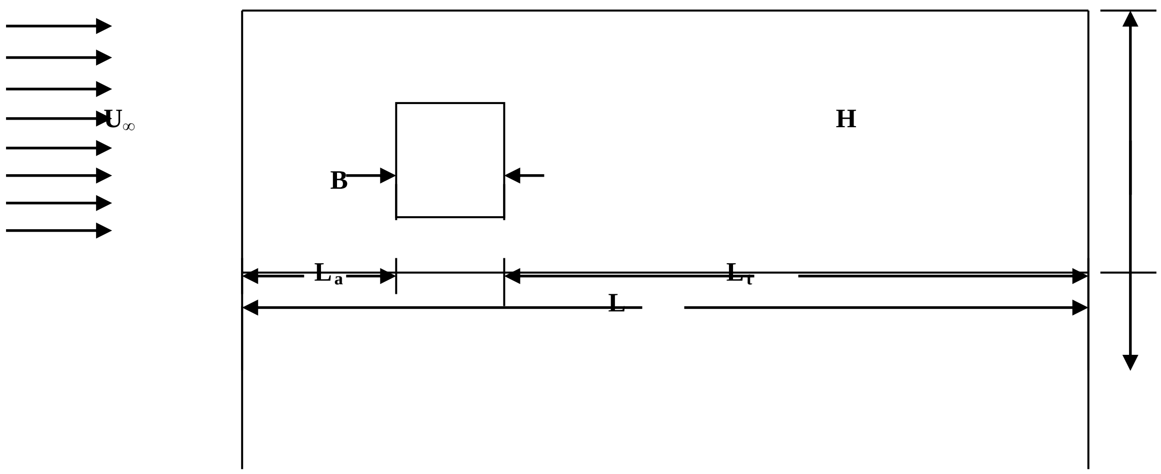

Fig 3.1 Geometrical model of flow configuration

Forced Convection			
L_a / B	L_t / B	L / B	H / B
8.5	16.5	26	20

Table 2.1 Computational Domain Parameters

2.2 Boundary Condition: Boundary conditions specify the flow and thermal variables on the boundaries of the physical model. They are, therefore, a critical component of simulations and it is important that they are specified appropriately. The computational domain uses following boundary conditions. The following boundary conditions are assigned in FLUENT.

Zone	Assigned Boundary Type
INLET	VELOCITY INLET
OUTLET	PRESSURE OUTLET
SQUARE CYLINDER	WALL(NO-SLIP)
TOP SURFACE	SYMMETRY
BOTTOM SURFACE	SYMMETRY

Boundary conditions (Table -3.2)

Inlet Boundary Condition
Since the flow is purely one dimensional hence no flow exists in y and z direction.
$u = u_{in}$, $v = w = 0$, $P_{inl} = P_{atm}$ = 1.03215 bar, u =0.0007338 m/s, $T_{atm} = T_\infty$ = 300 K

Outlet Boundary Condition

In fluent outlet condition is taken as pressure outlet.

Boundary Condition at the square Cylinder Surface
The no-slip boundary condition is applied on the square cylinder surface.

$$(u = v = w = 0),\ T = 400\ K$$

Boundary Condition at the Top and Bottom
The confining surfaces at y = ± H/2 are modeled as the symmetry condition.

2.3 Governing Equation:
The governing equations for this problem are the two dimensional continuity and Navier-Stokes momentum equations.
Continuity Equation.
This equation states that mass of a fluid is conserved.

Rate of increase of mass in element	=	Net rate of flow of mass fluid into fluid element

For time dependent.3-D equation is

$$\frac{\partial\rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \qquad (2.1)$$

For 2-D, incompressible and steady flow

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad (2.2)$$

X- Momentum Equation

Momentum equations are based on Newton's second law which states that, the rate of change of momentum equals the sum of forces on fluid particle. Time dependent and 3-d momentum in x-direction is

$$\frac{\partial(\rho u)}{\partial t}+u\frac{\partial(\rho u)}{\partial x}+v\frac{\partial(\rho u)}{\partial y}+w\frac{\partial(\rho u)}{\partial z}=-\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}[\lambda\nabla.V+2\mu\frac{\partial u}{\partial y}]+\frac{\partial}{\partial y}[\mu(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y})]+\frac{\partial}{\partial z}[\mu(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x})]+\rho f_x \qquad (2.3)$$

Where *V*= *ui*+*vj*+*wk* is velocity vector field, *f* denotes body force per unit mass, f_x as its x component and $\lambda=-\frac{2}{3}\mu$

For 2-D, incompressible, steady and with no body forces

$$\rho(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y})=-\frac{\partial p}{\partial x}+\mu(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}) \qquad (2.4)$$

Y- Momentum Equation

Time dependent and 3-d momentum in y-direction is

$$\frac{\partial(\rho v)}{\partial t}+u\frac{\partial(\rho v)}{\partial x}+v\frac{\partial(\rho v)}{\partial y}+w\frac{\partial(\rho v)}{\partial z}=-\frac{\partial p}{\partial y}+\frac{\partial}{\partial y}[\lambda\nabla.V+2\mu\frac{\partial v}{\partial y}]+\frac{\partial}{\partial x}[\mu(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y})]+\frac{\partial}{\partial z}[\mu(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y})]+\rho f_y$$
(2.5)

Where f_y denotes y-component of body force (*f*) per unit mass.

For 2-D, incompressible, steady and with no body forces

$$\rho(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y})=-\frac{\partial p}{\partial x}+\mu(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}) \qquad (2.6)$$

Energy Equation

$$\frac{\partial\theta}{\partial r}+\frac{\partial U\theta}{\partial x}+\frac{\partial V\theta}{\partial y}=\frac{1}{Re.Pr}(\frac{\partial^2\theta}{\partial X^2}+\frac{\partial^2\theta}{\partial Y^2}) \qquad (2.7)$$

With

$U=\frac{u}{u_\infty}$, $V=\frac{v}{u_\infty}$, $\tau=\frac{tu_\infty}{B}$, $X=\frac{x}{B}$, $Y=\frac{y}{B}$, $P=\frac{p}{\rho u_\infty^2}$, $\theta=\frac{T-T_\infty}{T_w-T_\infty}$

3. Discretised Equations for Solution Algorithm

It is common practice to subdivide the computational domain into number of sub domains or elements such that separate profile assumption can be made with each sub domain. In order to solve the non- linearity associated with fluid flow and energy equations the pressure-velocity linkage can be resolved by adopting an iterative solution strategy such as SIMPLE / SIMPLEC algorithm. A guessed pressure field is used to solve the momentum equation and pressure correction equation field which is in turn used to update the velocity and pressure field. To start the iteration process we use initial guess for velocity and pressure fields. As the algorithm proceeds our aim is to progressively to improve these guessed fields. The process is iterated until the convergence of velocity and pressure fields are obtained.

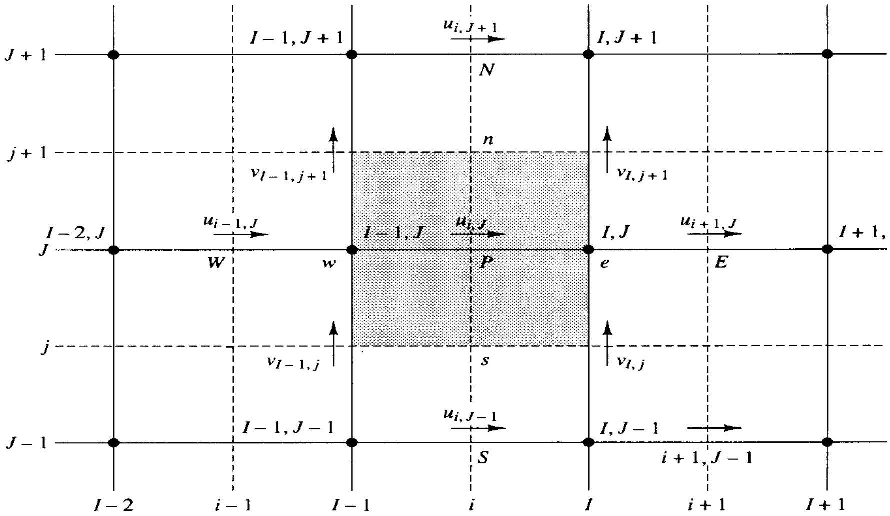

Fig. 3.1 Grid Arrangement for Flow Calculation in Two Dimensions

The discretised momentum equations in x direction:

$$a_{i,J}u_{i,J}=\sum a_{nb}u_{nb}+(P_{I-1,J}-P_{I,J})A_{i,J}+b_{i,J} \qquad (4.1)$$

The discretised momentum equation for y direction

$$a_{I,j}v_{I,j} = \sum a_{nb}v_{nb} + \left(P_{I,J-1} - P_{I,J}\right)A_{I,j} + b_{I,j} \quad (4.2)$$

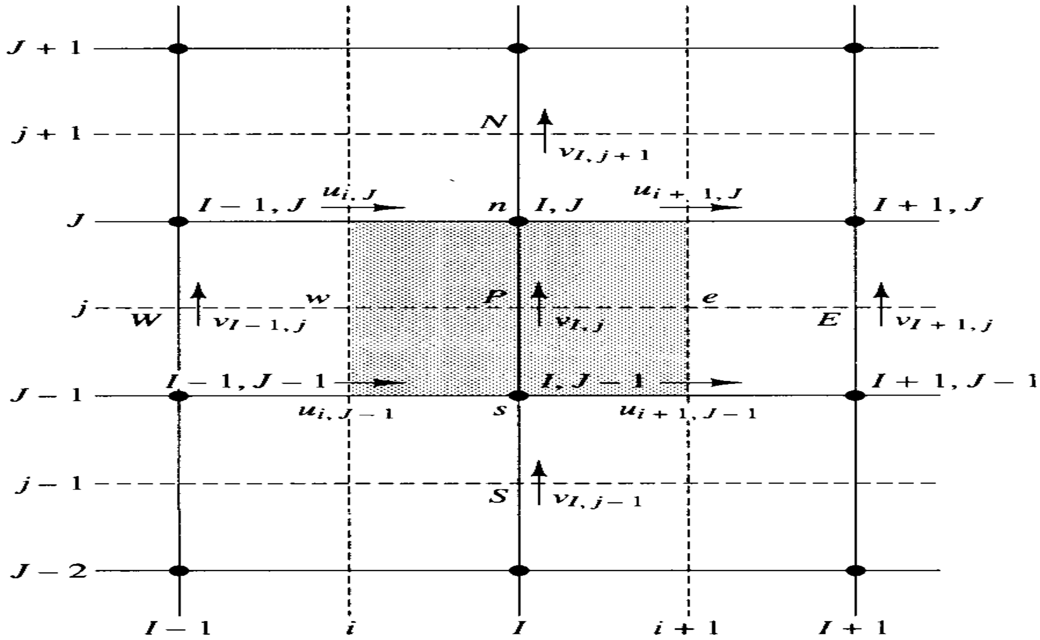

Fig. 3.2 Control Volume and Its Neighbouring Velocity Components

3.1 The SIMPLE Algorithm

The SIMPLE algorithm stands for Semi–implicit Method for Pressure Linked Equations. This is essentially a guess and correct procedure for the calculation of pressure on staggered grid for the discretised momentum equations. This pressure-velocity Coupling algorithm uses a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field.

$$a_{i,J}u_{i,J} = \sum a_{nb}u_{nb} + \left(P_{I-1,J} - P_{I,J}\right)A_{i,J} + b_{i,J}$$

This method can be explained with the two dimensional steady state laminar flow equations. The guessed pressure for the above equations is p* while the velocities are u* and v* as follows.

$$a_{i,J}u^{*}_{i,J} = \sum a_{nb}u^{*}_{nb} + \left(P^{*}_{I-1,J} - P^{*}_{I,J}\right)A_{i,J} + b_{i,J} \quad (3.1)$$

$$a_{I,j}v^{*}_{I,j} = \sum a_{nb}v^{*}_{nb} + \left(P^{*}_{I,J-1} - P^{*}_{I,J}\right)A_{I,j} + b_{I,j} \quad (3.2)$$

Now the correction p', u' and v' may be introduced as (correction formulae)

$$p = p^{*} + p' \quad (3.3)$$

$$u = u^{*} + u' \quad (3.4)$$

$$v = v^{*} + v' \quad (3.5)$$

Where P=correct pressure field and P^{*} is =guessed pressure field.

Substitution of correct pressure field p into momentum equations yield correct velocity field.

Subtraction of equations (4.1) and (4.2) from (4.3) and (4.4) respectively would give us

$$a_{i,J}\left(u_{i,J} - u^{*}_{i,J}\right) = \sum a_{nb}\left(u_{nb} - u^{*}_{nb}\right) + \left[\left(p_{I-1,J} - p^{*}_{I-1,J}\right) - \left(p_{I,J} - p^{*}_{I,J}\right)\right]A_{i,J} \quad (3.6)$$

$$a_{i,J}\left(v_{i,J} - v^{*}_{i,J}\right) = \sum a_{nb}\left(v_{nb} - v^{*}_{nb}\right) + \left[\left(p_{I,J-1} - p^{*}_{I,J-1}\right) - \left(p_{I,J} - p^{*}_{I,J}\right)\right]A_{I,j} \quad (3.7)$$

Using correction formulas the equation (4.8) and (4.9) may be written as:

$$a_{i,J}u'_{i,J} = \sum a_{nb}u'_{nb} + \left(p'_{I-1,J} - p'_{I,J}\right)A_{i,J} \quad (3.8)$$

$$a_{i,j}v'_{i,j} = \sum a_{nb}v'_{nb} + \left(p'_{I,J-1} - p'_{I,J}\right)A_{I,j} \quad (3.9)$$

In order to simplify the above equations the two approximations

$$\sum a_{nb}u'_{nb} \quad and \quad \sum a_{nb}v'_{nb}$$

are dropped. The omissions of these terms are the main approximations of SIMPLE algorithm. We obtained

$$u'_{i,J} = d_{i,J}\left(p'_{I-1,J} - p'_{I,J}\right) \quad (3.10)$$

$$v'_{I,j} = d_{I,j}\left(p'_{I,J-1} - p'_{I,J}\right) \quad (3.11)$$

Where $d_{i,j} = \frac{A_{i,j}}{a_{i,j}}$ and $d_{I,j} = \frac{A_{I,j}}{a_{I,j}}$

So far we have considered momentum equations but velocity field also subjected to constraint that it should also satisfy continuity equations. The continuity equation for the control volume is

$$\left[(\rho uA)_{i+1,J} - (\rho uA)_{i,J}\right] + \left[(\rho vA)_{I,j+1} - (\rho vA)_{I,j}\right] = 0 \qquad (3.12)$$

Substitution of corrected velocities of equations into discretised continuity equations gives:

$$\left[\rho_{i+1,J} A_{i+1,J}\left(u^*_{i+1,J} + d_{i+1,J}\left(p'_{I,J} - p'_{I+1,j}\right)\right) - \rho_{i,J} A_{i,J}\left(u^*_{i,j} + d_{i,J}\left(p'_{I-1,J} - p'_{I,J}\right)\right)\right]$$
$$+ \left[\rho_{I,j+1} A_{i,J+1}\left(v^*_{I,j+1} + d_{I,j+1}\left(p'_{I,J} - p'_{I,J+1}\right)\right) - \rho_{I,j} A^*_{I,j}\left(v^*_{I,j} + d_{I,j}\left(p'_{I,J-1} - p'_{I,J}\right)\right)\right] = 0$$

Identifying the coefficient of p' it may be written as

$$a_{I,J} p'_{I,J} = a_{I+1,J}\ p'_{I+1,J} + a_{I-1,J}\, p'_{I-1,J} + a_{I,J+1} p'_{I,J+1} + a_{I,J-1} p'_{I,J-1} + b'_{I,J} \qquad (3.13)$$

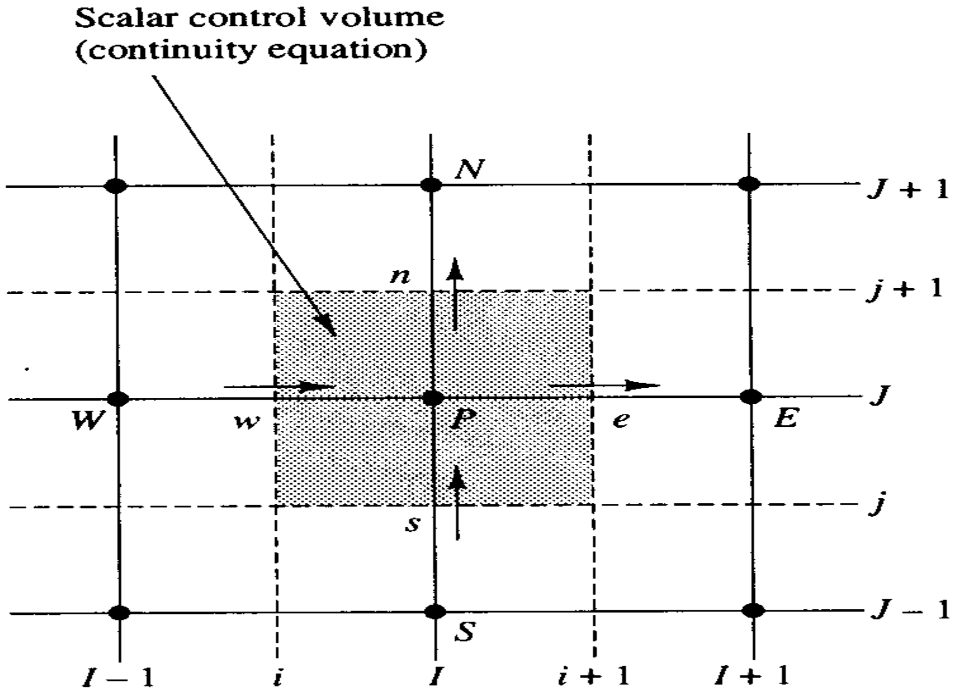

Fig. 4.6 Scaler Control Volume Used For Discretisation of Continuity equation

Where

$a_{I+1,J}$	$a_{I-1,J}$	$a_{I,J+1}$	$a_{I,J-1}$	$b'_{I,J}$
$(\rho dA)_{i+1,J}$	$(\rho dA)_{i,J}$	$(\rho dA)_{I,j+1}$	$(\rho dA)_{i,J}$	$(\rho u^* A)_{i,J} - (\rho u^* A)_{i+1,J} + (\rho v^* A)_{I,j} - (\rho v^* A)_{I,j+1}$

The above equation (4.16) represents the discretised continuity equation as an equation for pressure correction p'. By solving above pressure correction equation the correct pressure field may be known and correspondingly substitute pressure field into continuity equation would give us the correct velocity field. The omission of the terms such as $\sum a_{nb} u'_{nb}$ in the derivation does not affect much the final results. Because the pressure correction and velocity corrections will be zero in a converged solution giving p*=p and u*=u.

3.2 Sequence of Operation for the SIMPLE Algorithm

1: Guess the pressure filed p*
2: Solve the momentum equations to obtain u* and v*
3: Solve the p' equation
4: Calculate the p by adding p' to p*
5: Calculate the u and v from velocity correction formula.
6: Treat the corrected pressure p as new guessed pressure p*.return to
Step 2 and repeat the whole procedure until a converged solution is obtained.

3.3 The SIMPLE Algorithm

The SIMPLE algorithm follows the same step as he SIMPLE algorithm, with the difference that the momentum equations are manipulated so that the SIMPLEC velocity correction equations omit terms that are less significant than those omitted in simple.

The u velocity correction equation of SIMPLEC is given by

$$u'_{i,J} = d_{i,J}\left(p'_{I-1,J} - p'_{I,J}\right)$$

Where

$$d_{i,j} = \frac{A_{i,j}}{a_{i,j} - \sum a_{nb}}$$

Similarly the modified v- velocity correction equation is

$$v'_{I,j} = d_{I,j}\left(p'_{I,J-1} - p'_{I,J}\right)$$

Where

$$d_{I,j} = \frac{A_{I,j}}{a_{I,j} - \sum a_{nb}}$$

The discretised pressure correction equations are same the same as in SIMPLE except that the d-terms .the sequence of operations of the SIMPLEC algorithm is identical to that of SIMPLE.

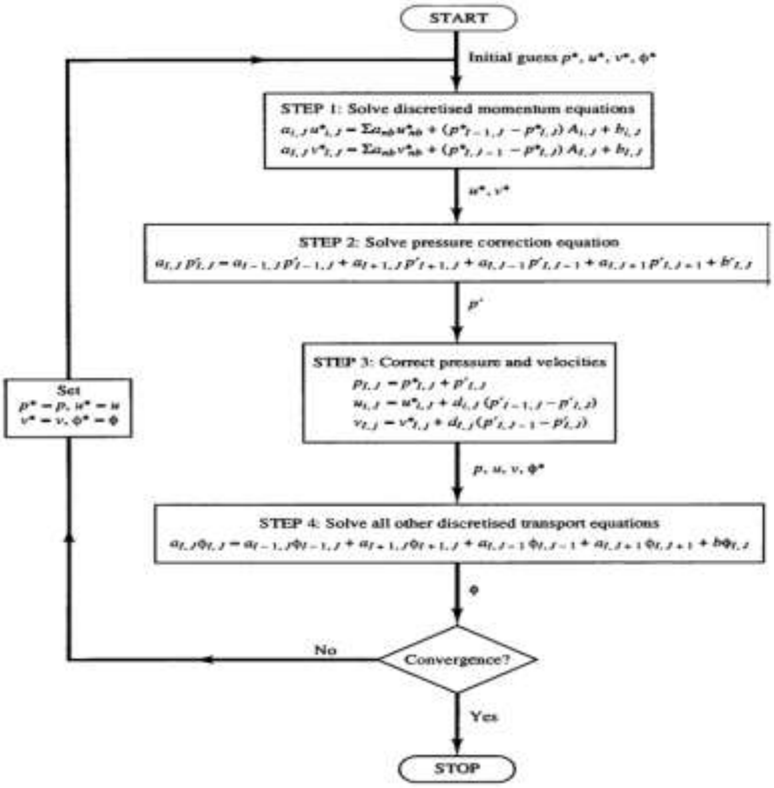

Fig. 4.7 Flow chart for simple algorithm

4. RESULTS AND DISCUSSION

In the present study, two dimensional numerical simulation of flow past a square cylinder has been carried out for various cylinder width (B=1, 2 & 3) & Reynolds number and results are compared with the experimental and numerical data available in the literature. The flow features are represented with the help of Recirculation Length.

The recirculation length is defined as the stream-wise distance from the trailing end of the square cylinder to the re-attachment point along the wake centerline. The location of the re-attachment point is determined computationally by monitoring the streamwise velocity along the streamwise centerline of the cylinder and moving downstream till it changes its sign from negative to positive. Figures 5.1, 5.2 & 5.3 show the computed values of recirculation length for the square cylinder for various cylinder widths (B=1, 2 & 3) compared with Atul et al. (2000) is:

RL=0.0672 x Re

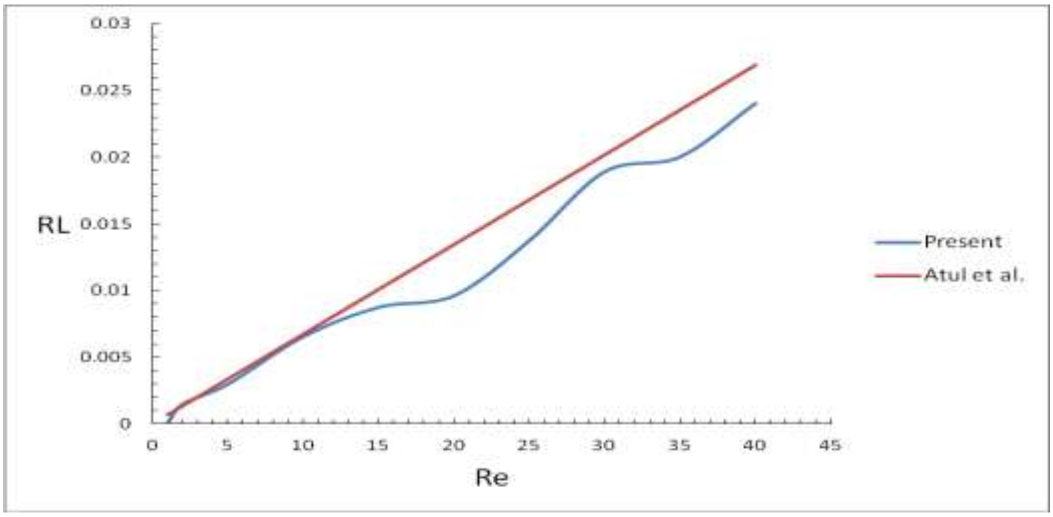

Fig. 5.1 Variation of Recirculation Length at B=1

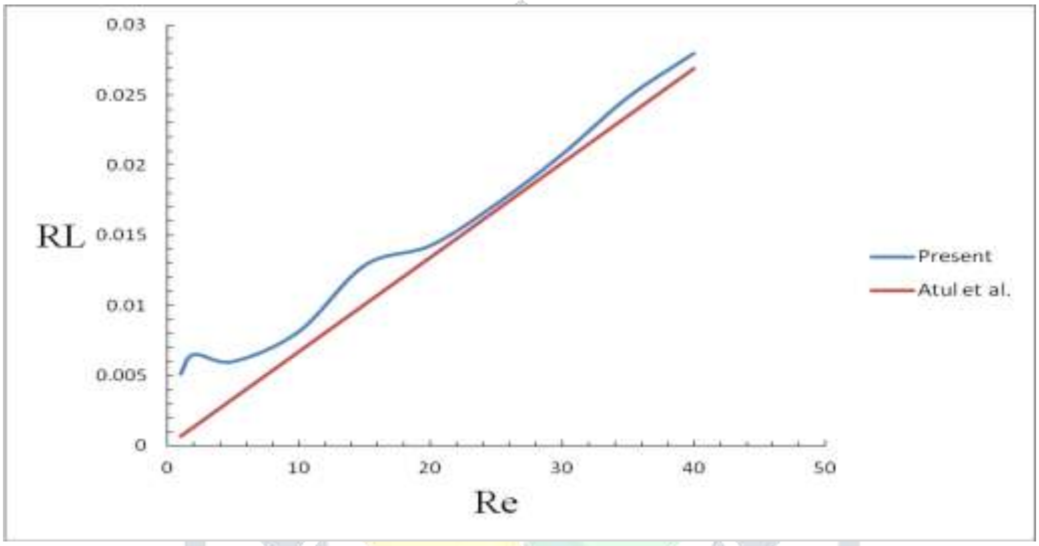

Fig. 5.2 Variation of Recirculation Length at B=2

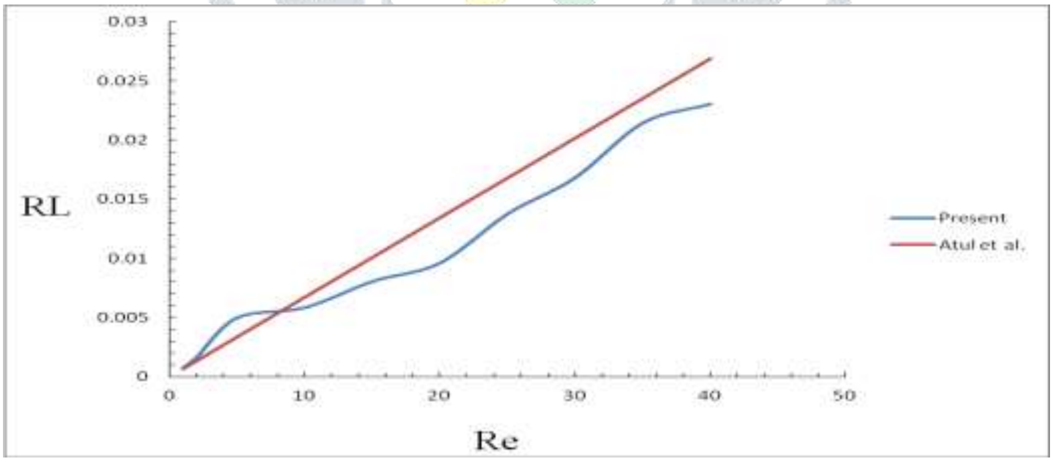

Fig. 5.3 Variation of Recirculation Length at B=3

From the figure 5.1 it is cleared that the present value of Recirculation length is approaches to the calculated value from Atul et al. (2000) and whereas in case of B=2 & 3 the present value is greater or lower respectively.

4.1 Conclusion

Numerical investigation are conducted on unconfined flow past a square cylinder with forced convection heat transfer for different cylinder widths (B=1, 2 & 3) and $1 \leq Re \leq 160$. As the cylinder width and Reynolds number changes, a significant change in the properties of flow is observed. The flow is steady for $Re \leq 40$ and become unsteady when $Re \geq 50$ and transition occurs at $40 \leq Re \leq 50$. The sequence of different events involved in shedding at $Re \geq 100$ is shown clearly.

The correlation between non dimensional recirculation length and Reynolds number has been shown for $1 \leq Re \leq 40$ in the steady flow regime. The result show good match with the available numerical and experimental data. As the Reynolds number increase the recirculation length increases behind the cylinder. Change in the cylinder width also affects the recirculation length. As the cylinder width increases the length of recirculation increases.

REFERENCES

[1] Abernathy, F.H. and Kronauer, R.E., (1961), The Formation of Vortex Sheets, *J. Fluid Mech*., Vol.13, pp. 1-20.

[2] Aung, W.,(1987), Mixed Convection in Internal flow, in S.Kakac, R.K. Shah and W.Aung, Eds., Handbook of Single Phase Convection Heat transfer, Wiley-Interscience, New York, Chap.15.

[3] Badr, H.M., (1984), A Theoretical study of Laminar Mixed Convection from a Horizontal Cylinder in a cross stream, *Int. J. Heat Transfer*, Vol. 26, pp. 639-653.

[4] Badr, H.M., (1984), Laminar Combined Convection from a Horizontal Cylinder-Parallel and Contra flow Regimes, *Int. J. Heat Transfer* Vol.27, pp. 15-27.

[5] Breuer, M. Bernsdorf, J., Zeiser, T. and Durst, F., (2000), Accurate Computations of the Laminar flow Past a Square Cylinder based on two different Methods : Lattice Boltzmann and Finite Volume , *Int. J. Head Fluid Flow*, Vol.21, pp. 186-196

[6] Chang, K.S. AND Sa, J.Y., (1990), The effect of Vortex Shedding in the near wake of a Circular Cylinder , *J.Fluid Mech*., Vol. 220 , pp. 253-266.

[7] Davis, R.W., Morre, E.F. and Purtell, L.P., (1984), a Numerical Study of Confined Flow around Rectangular Cylinders, Phys. Fluids, Vol. 27, pp. 46-59.

[8] Davis, R.W., Morre, E.F., 1982), A Numerical Study of Vortex Shedding from Rectangles , *J. Fluid Mech*., Vol. 116, pp. 475-506.

[9] Dennis, S.C.R., Hudson , J.D. and Smith, N., (1968) , Steady Laminar Forced Convection from a Circular Cylinder at Low Reynolds Number., Phys. Fluid, Vol. 11, pp. 933-940.

[10] Devarkonda, R., (1994), Experimental and Numerical Investigation of Un-steady Bluff Body flows, Ph.D. Thesis, University of California, Berkeley

[11] Farouk, B. and Guceri, S.I., (1982), Natural and Mixed Convection Heat Transfer around A Horizontal Cylinder Within Confining Walls, Numerical Heat Transfer, Vol. 5, pp. 329-341

[12] Hatanaka, K. and Kawahara, M., (1995), A Numerical Study of Vortex Shedding around a Heated/Cooled Circular Cylinder by the Three- Step Taylor-Gaylerkin Method, *Int. J. Numer. Meth. Fluids*, Vol. 21, pp. 857-867