EFFECT OF CYLINDER WIDTH AND REYNOLS NUMBER ON RECIRCULATION LENGTH OF UNCONFINED FLOW PAST A SQUARE CYLINDER WITH FORCED CONVECTION HEAT TRANSFER

[1]Avinash Dholiwal, [2]Sandeep Phogat
[1]Assistant Professor, [2]Assistant Professor
[1]Mechanical Engineering Department, [2]Mechanical Engineering Department,
[1]Amity University Haryana, Gurgaon, Haryana, [2] Amity University Haryana, Gurgaon, Haryana.

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 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.

Index Terms **– Reynolds Number, Recirculation Length, Velocity Profile**

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.

When a fluid particle flows within the boundary layer around the circular cylinder, it has been observed experimentally that, the pressure is maximum at the stagnation point and gradually decreases along the front half of the cylinder. The flow stays attached in this favorable pressure region as expected. However, the pressure starts to increase in the rear half of the cylinder and at this time particle experiences an adverse pressure gradient. Consequently, the flow separates from the surface, creating a highly turbulent region behind the cylinder called the wake. The pressure inside the wake region remains low as the flow separates and a net pressure force (pressure drag) is produced.

2. PROBLEM FORMULATION

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

1. B = width of square cylinder
2. L = length of domain
3. L_a = distance between the inlet and front surface of square cylinder
4. L_t = distance between the exit and rear surface of square cylinder
5. H = height of the domain.

3. 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

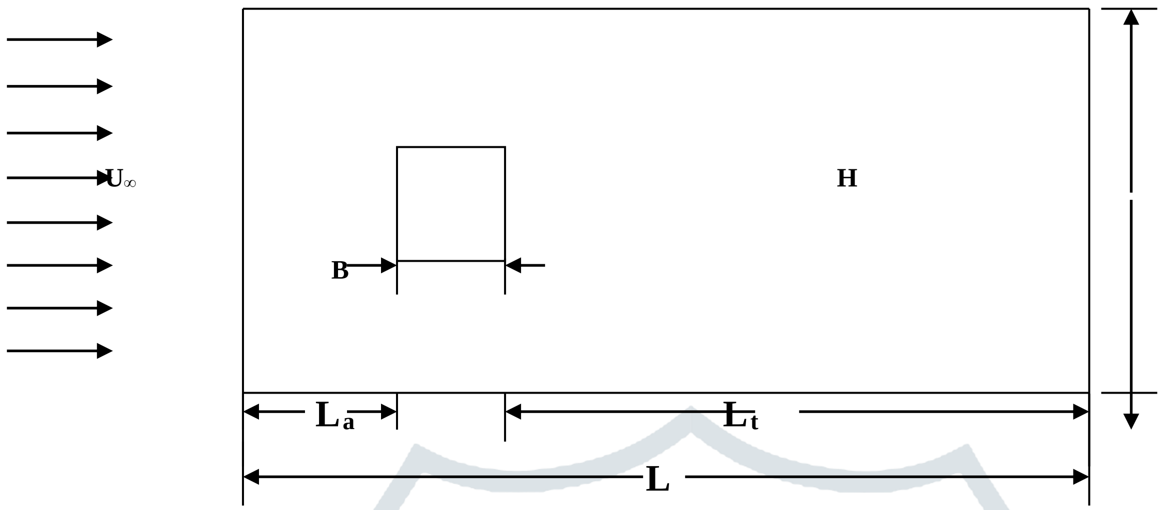

Fig 3.1 Geometrical model of flow configuration

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

Table 3.1 Computational Domain Parameters

3.1 Grid System

We are dividing the computational domain into a set of rectangular cells and a staggered grid arrangement, shown in Fig. 3.2, is used such that the velocity components are defined at the center of the cell faces to which they are normal. The pressure is defined at the center of the cell. The grids are non uniform on the horizontal *x*-y plane. The grids are clustered near the obstacle to resolve the boundary layer. One such grid on the two dimensional plane *(x*- y plane) is illustrated in Fig. 3.3.

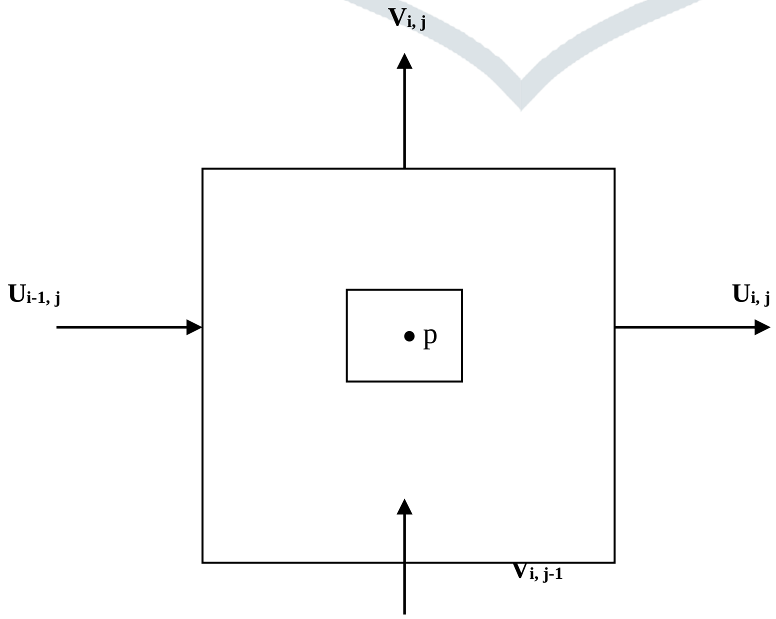

Fig.3.2: Staggered grid arrangement

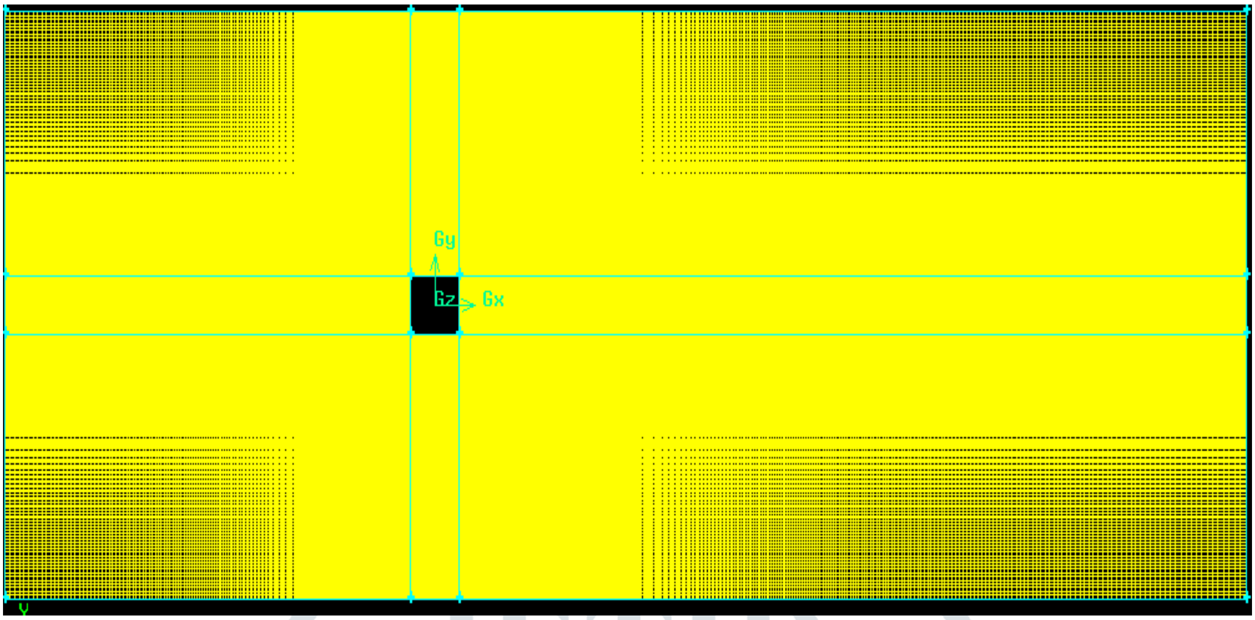

Fig.3.3 Meshing of the flow pattern (B=1)

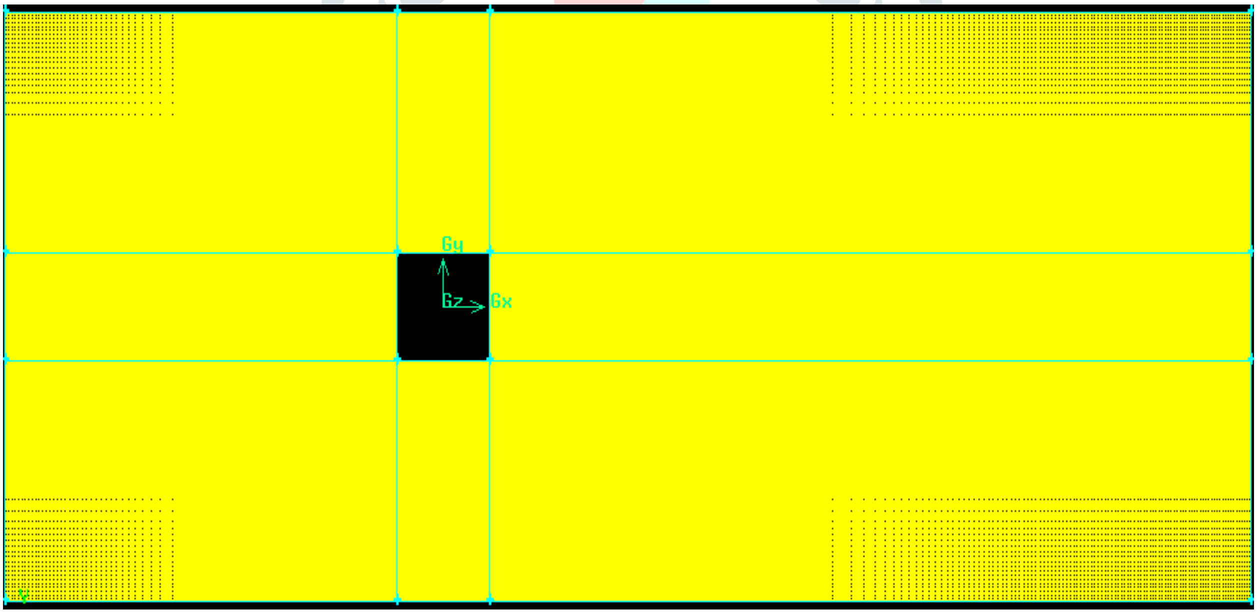

Fig.3.4 Meshing of the flow pattern (B=2)

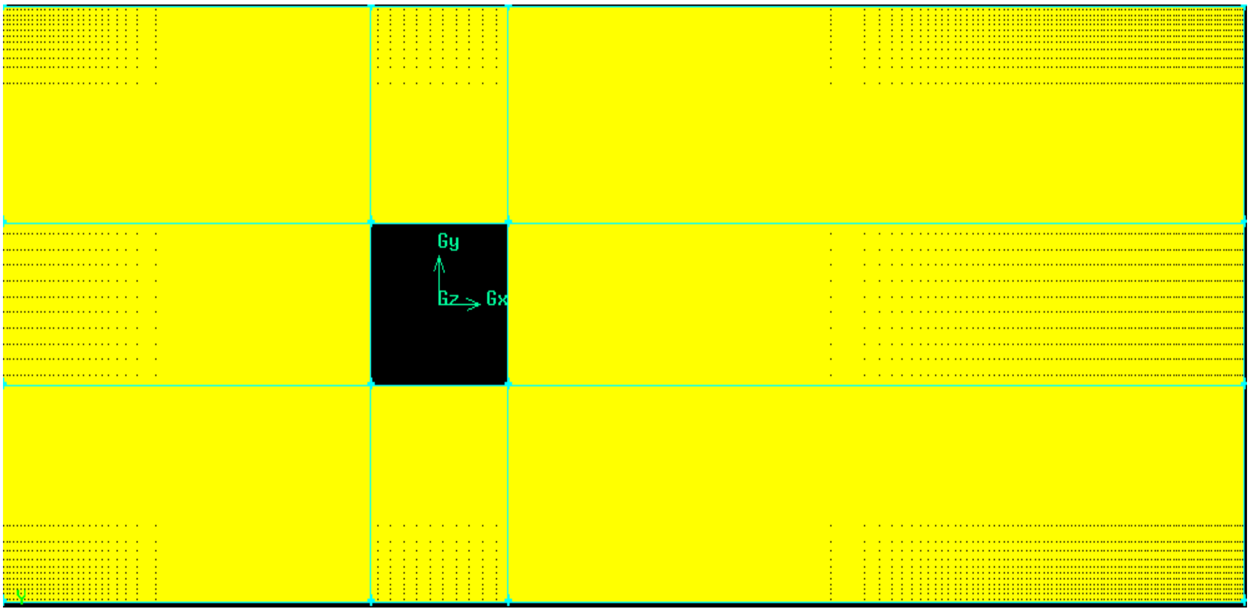

Fig.3.5 Meshing of the flow pattern (B=3)

3.2 Assumptions

A two-dimensional flow is considered in this thesis because it simplifies the problem, both mathematically and computationally. The temperature of cylinder is assumed to be constant (400 K) which is appropriate since we are considering the cylinder as thermal sources or sinks. Following assumptions are considered:

(a). Flow is two-dimensional and laminar.

(b). The fluid is air which is considered incompressible and Newtonian

(c). Viscous dissipation and compression work is negligible.

(d). Surface of the square cylinder is smooth.

(e). Radiative heat transfer is insignificant.

3.3 Boundary Conditions

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.

3.4 **Governing Equations**

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

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \quad (3.1)$$

For 2-D, incompressible and steady flow

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (3.2)$$

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

$$\frac{\partial(\rho u)}{\partial t} + u\frac{\partial(\rho u)}{\partial x} + v\frac{\partial(\rho u)}{\partial y} + w\frac{\partial(\rho u)}{\partial z} = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}[\lambda \nabla . V + 2\mu\frac{\partial u}{\partial y}] + \frac{\partial}{\partial y}[\mu(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y})] + \frac{\partial}{\partial z}[\mu(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x})] + \rho f_x \quad (3.3)$$

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

$$\rho(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}) = -\frac{\partial p}{\partial x} + \mu(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}) \quad (3.4)$$

Y- Momentum Equation

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

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

(3.5)

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

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

$$\rho(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y}) = -\frac{\partial p}{\partial x} + \mu(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}) \quad (3.6)$$

Energy Equation

$$\frac{\partial \theta}{\partial r} + \frac{\partial U\theta}{\partial x} + \frac{\partial V\theta}{\partial y} = \frac{1}{Re.Pr}(\frac{\partial^2 \theta}{\partial X^2} + \frac{\partial^2 \theta}{\partial Y^2}) \quad (3.7)$$

With

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

.

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.

4.1 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 stream wise velocity along the stream wise 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

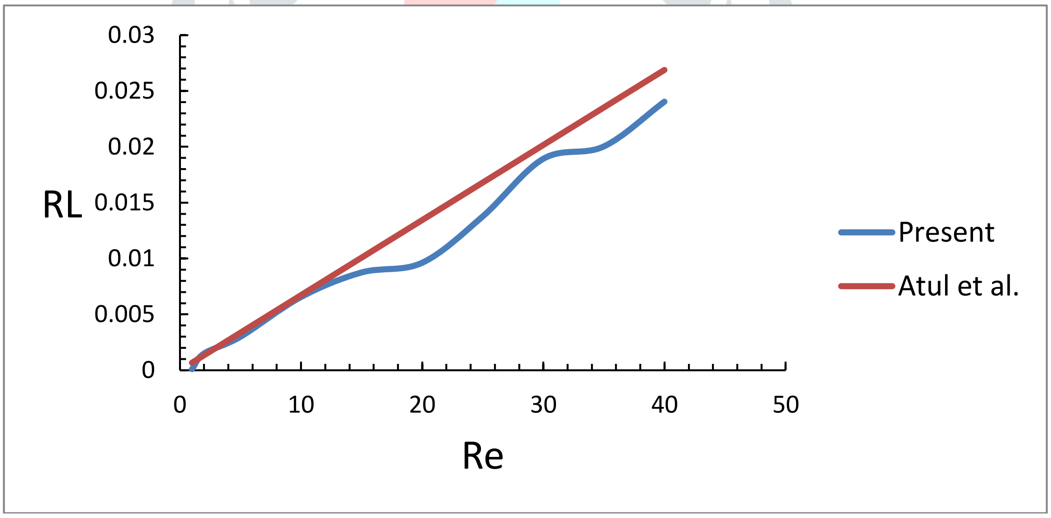

Fig. 4.1 Variation of Recirculation Length at B=1

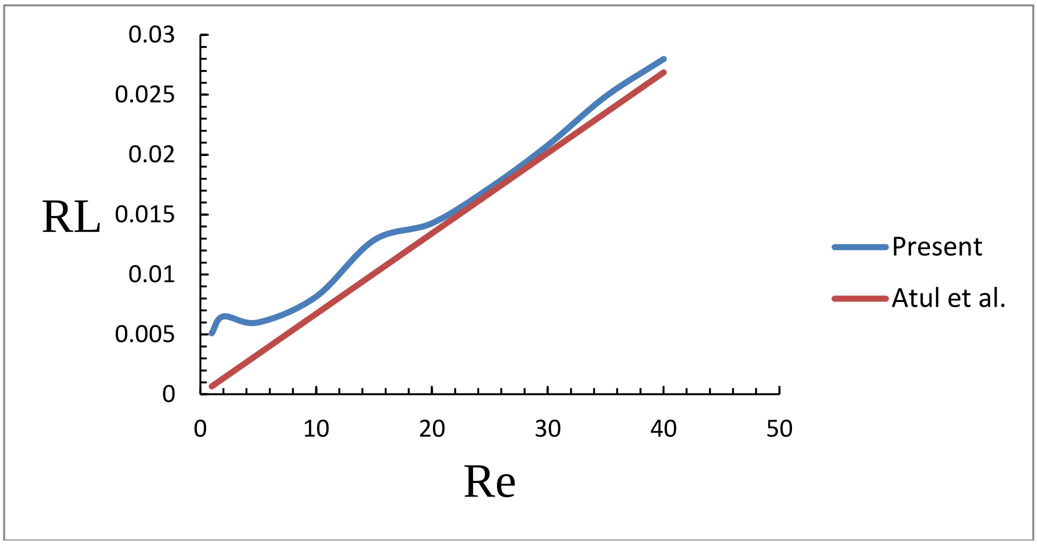

Fig. 4.2 Variation of Recirculation Length at B=2

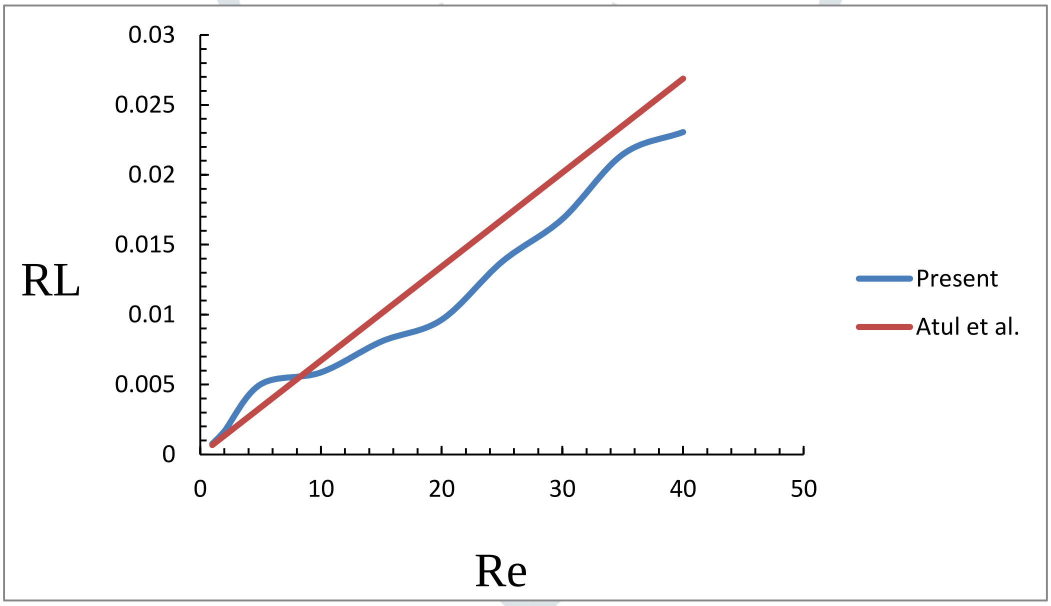

Fig. 4.3 Variation of Recirculation Length at B=3

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.

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