EFFECT OF CYLINDER WIDTH AND REYNOLS NO ON VELOCITY PROFILE OF UNCONFINED FLOW PAST A SQUARE CYLINDER WITH FORCED CONVECTION HEAT TRANSFER

[1]Avinash Dholiwal

[1]Assistant Professor
[1]Mechanical Engineering
[1]Amity University Haryana, Gurgaon, 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 paper 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**.**

***Index Terms* – Reynolds No., Heat Transfer, Cylinder Width.**

I. INTRODUCTION

A body placed in the flow field is of considerable interest as both the flow field and body interact with each other. Several problems involving the flow of fluid around submerged objects are encountered in the various engineering fields. Such problems may have either a fluid flowing around a stationary submerged object, or an object moving through a large mass of stationary fluid, or both the object and the fluid are in motion. Knowledge of forces exerted by the fluid on object is of significant importance in their design and analysis. The force exerted by the fluid on a moving body or on a stationary body by fluid in motion can be resolved into two components, one in the direction of motion and other perpendicular to the direction of motion. The component parallel to the flow is called viscous drag and is due to the shear stress on the surface. The component perpendicular to the direction of motion of the flow is called pressure drag. This pressure drag tries to lift the body. However for a symmetric body, such as for a sphere or a cylinder, facing the flow symmetrically, there is no lift force and thus the total force exerted by the fluid is equal to the drag on the body. The flow past a body is of direct relevance to the design of structures, heat exchanger components and where even flow induced vibration is important. The analysis of flow past a body in non-uniform stream is more complex. The approaching flow of a curved river against the bridge pier is one such example. Submarines, ships, aircraft, automobiles and missiles are examples where the object is in motion and the fluid is stationary. High rise buildings, chimneys and tube banks of heat exchangers are examples where the fluid is in motion. It is important to know the velocity, pressure and temperature fields in detail in a large number of applications involving fluids, namely, liquid and gases. The performance of devices such as turbo-machinery and heat exchangers is determined entirely by the fluid motion within them and hence it is essential to know the pressure and velocity distribution to determine the effect on the body. The detailed nature of fluid flow over a square cylinder is one of the fundamental topics in classical fluid dynamics as it demonstrates flow separation and vortex shedding. At very low Reynolds numbers, the flow is steady and symmetrical. As the Reynolds number is increased, asymmetries and time-dependence develop, eventually resulting in the famous Von Karman vortex street, and then on to turbulence.

1.1 How Does a CFD Code Work?

CFD codes are structured around the numerical algorithms that tackle fluid and heat transfer problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces input problem parameters and to examine the results. Hence all codes contain three main elements:

1 Pre-processing.
2 Solver
3 Post-processing.

2. PROBLEM FORMULATION

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

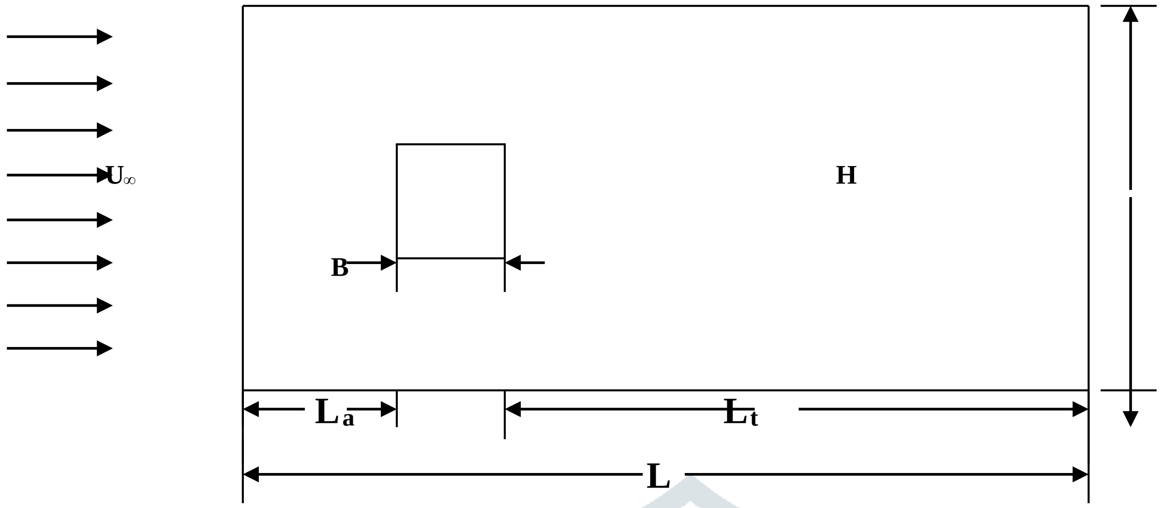


Fig 2.1 Geometrical model of flow configuration

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 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

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 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

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

For 2-D, incompressible and steady flow

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{2.2}$$

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

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

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

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

Y- Momentum Equation

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

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

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

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

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

Energy Equation

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

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. SIMPLE ALGORITHEM

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.

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

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.

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

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

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

$$p = p^{*} + p' \tag{3.3}$$

$$u = u^{*} + u' \tag{3.4}$$

$$v = v^{*} + v' \tag{3.5}$$

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

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

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

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

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

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

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

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

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

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

$$\left[(\rho uA)_{i+1,J} - (\rho uA)_{i,J}\right] + \left[(\rho vA)_{I,j+1} - (\rho vA)_{I,j}\right] = 0 \tag{4.14}$$

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

$$a_{I,J}p'_{I,J} = a_{I+1,J}\,p'_{I+1,J} + a_{I-1,J}p'_{I-1,J} + a_{I,J+1}p'_{I,J+1} + a_{I,J-1}p'_{I,J-1} + b'_{I,J} \tag{3.12}$$

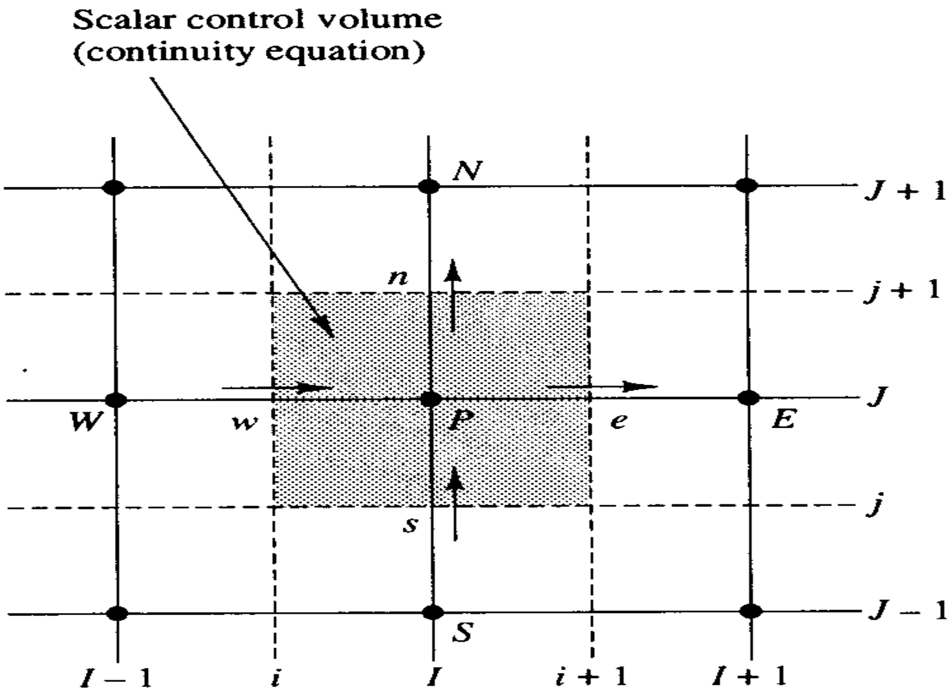


Fig. 3.1 Scaler Control Volume Used For Discretisation of Continuity equation

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

3.1 Flow Chart for Simple Algorithem

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

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

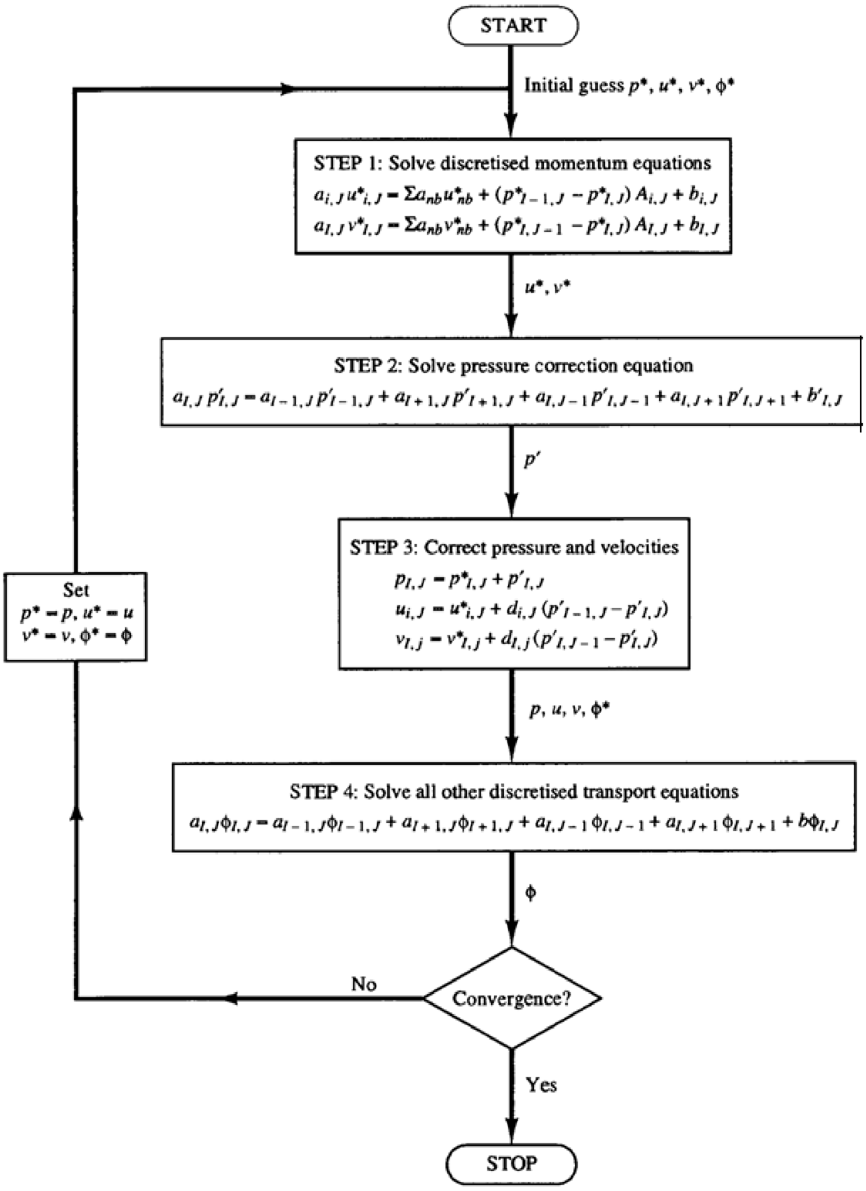
Similarly the modified v- velocity correction equation is

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

Where

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

The discretise 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.


START
Initial guess p^*, u^*, v^*, ϕ^*
STEP 1: Solve discretised momentum equations
$a_{i,J} u^*_{i,J} = \Sigma a_{nb} u^*_{nb} + (p^*_{I-1,J} - p^*_{I,J}) A_{i,J} + b_{i,J}$
$a_{I,J} v^*_{I,J} = \Sigma a_{nb} v^*_{nb} + (p^*_{I,J-1} - p^*_{I,J}) A_{I,J} + b_{I,J}$
u^*, v^*
STEP 2: Solve pressure correction equation
$a_{I,J} p'_{I,J} = a_{I-1,J} p'_{I-1,J} + a_{I+1,J} p'_{I+1,J} + a_{I,J-1} p'_{I,J-1} + a_{I,J+1} p'_{I,J+1} + b'_{I,J}$
p'
STEP 3: Correct pressure and velocities
$p_{I,J} = p^*_{I,J} + p'_{I,J}$
$u_{i,J} = u^*_{i,J} + d_{i,J} (p'_{I-1,J} - p'_{I,J})$
$v_{I,j} = v^*_{I,j} + d_{I,j} (p'_{I,J-1} - p'_{I,J})$
p, u, v, ϕ^*
STEP 4: Solve all other discretised transport equations
$a_{I,J} \phi_{I,J} = a_{I-1,J} \phi_{I-1,J} + a_{I+1,J} \phi_{I+1,J} + a_{I,J-1} \phi_{I,J-1} + a_{I,J+1} \phi_{I,J+1} + b_{\phi I,J}$
ϕ
Convergence?
No
Yes
STOP
Set
$p^* = p, u^* = u$
$v^* = v, \phi^* = \phi$

4. RESULT AND DISCUSSION

In this paper, for various cylinder width & Reynolds number, a two dimensional numerical simulation of flow past a square cylinder has been carried out and results are compared with experimental and numerical data. The flow features are represented with the help of Velocity Profile.

The stream line velocity at the inlet is same for all cylinder width for the same Re number for unconfined flow. The velocity of the fluid accelerates with increasing cylinder width. This is shown in following figures 4.1, 4.2 & 4.3 for the instantaneous cross sectional U-velocity profiles at three different x-locations (leading, centre and trailing) of square cylinder for various cylinder widths. As the fluid passes over the square cylinder, it shows appreciable change in velocity profile at leading, centre, trailing location for different cylinder width as shown below in the figures.

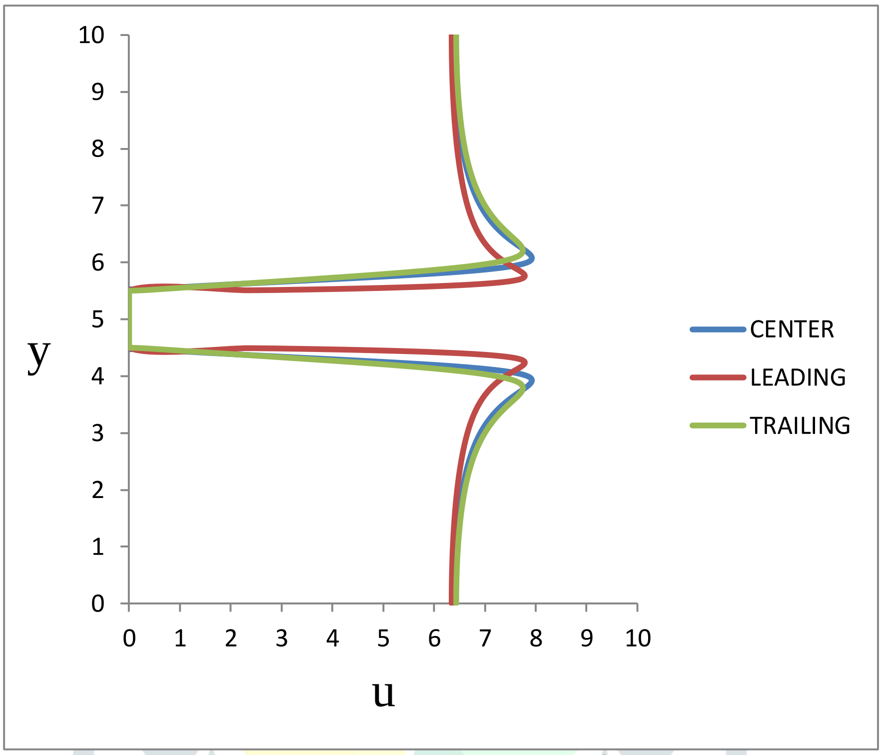


Fig. 4.1 Instantaneous cross-sectional U-velocity profiles (B=1, Re=40)

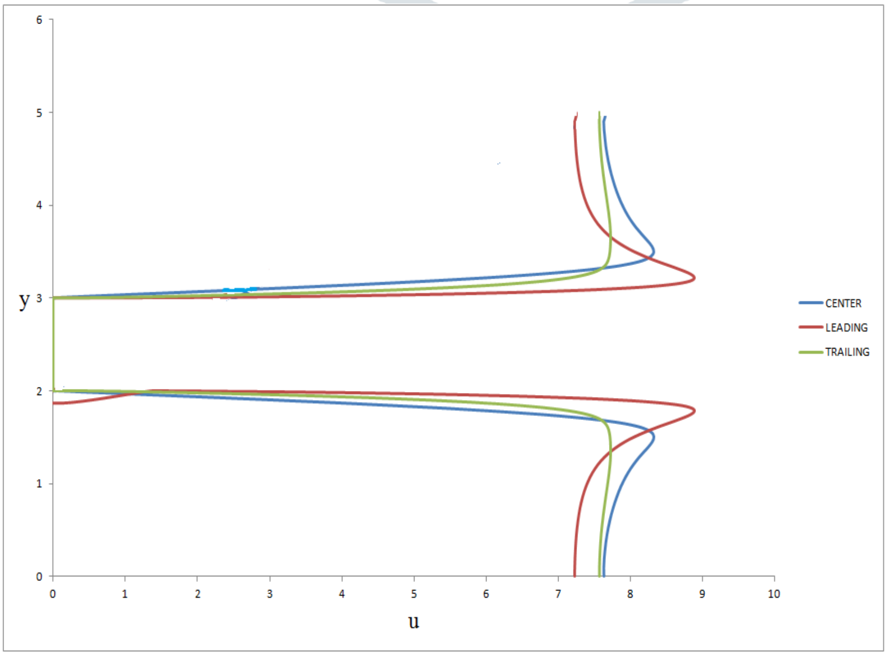


Fig. 4.2 Instantaneous cross-sectional U-velocity profiles (B=2, Re=100)

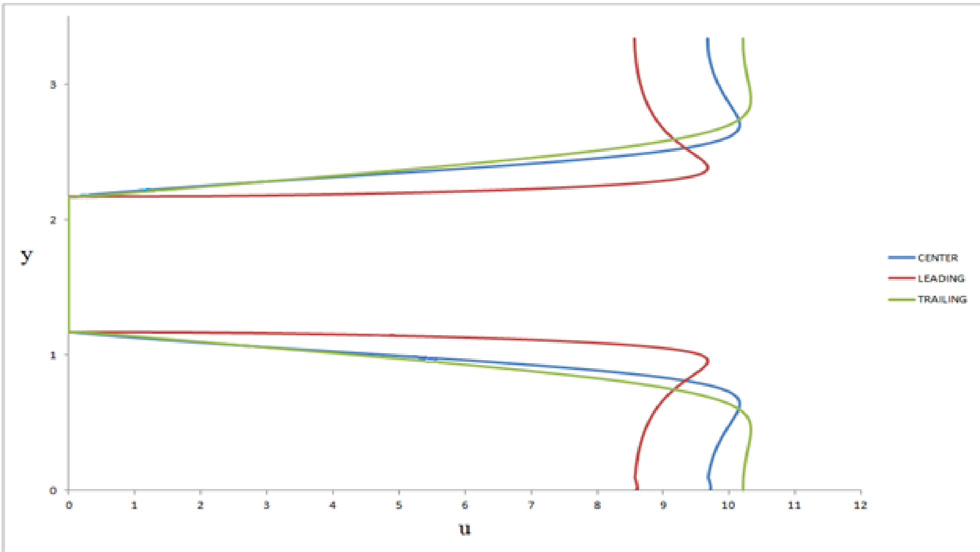


Fig. 4.3 Instantaneous cross-sectional U-velocity profiles (B=3, Re=150)

5. 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 effect of change in cylinder width and Reynolds number is considered on velocity profile for different values of cylinder widths (B=1, 2 & 3) and $1 < Re < 160$.

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