TWO-DIMENSIONAL NUMERICAL SIMULATION OF FLOW PAST A SQUARE CYLINDER

[1]Avinash Dholiwal

[1]Assistant Professor
[1] Mechanical Engineering Department,
[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 conditions 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 discretize 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 the literature. The result shows reasonably good matching.

Index Terms – **Recirculation Length, Simulation, Isotherm.**

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 crisscross-section presents an idealization of several industrially important applications. It is readily acknowledged that a systematic study of the flow past a single-cylinder only provides valuable insights into the nature of the 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.

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 vus 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 the other another perpendicular 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 Pre-Processing

This is the first step in building and analyzing a flow model. Preprocessor consists of input of a flow problem by means of an operator-friendly interface and subsequent transformation of this input into suitable form for the use of solver. The user activities at the Preprocessing stage involve:

1.   Definition of the geometry of the region: The computational domain.

2.   Grid generation the subdivision of the domain into a number of smaller, non-overlapping subdomains(or control volumes or elements Selection of physical or chemical phenomena that need to be modeled).

3.   Definition of fluid properties.

4.   Specification of appropriate boundary conditions at cells, which coincide with or touch the boundary.

1.2 Solver

The CFD solver does the flow calculations and produces the results. The numerical solution of Navier–Stokes equations in CFD codes usually implies a discretization method: it means that derivatives in partial differential equations are approximated by algebraic expressions which can be alternatively obtained by means of the finite-difference or the finite-element method. Otherwise, in a way that is completely different from the previous one, the discretization equations can be derived from the integral form of the conservation equations: this approach, known as the finite volume method, is implemented in FLUENT, because of its adaptability to a wide variety of grid structures. The result is a set of algebraic equations through which mass, momentum, and energy transport are predicted at discrete points in the domain. In the freeboard model that is being described, the segregated solver has been chosen so the governing equations are solved sequentially. Because the governing equations are non-linear and coupled, several iterations of the solution loop must be performed before a converged solution is obtained and each of the iteration is carried out as follows:

(1).Fluid properties are updated in relation to the current solution; if the calculation is at the first iteration, the fluid properties are updated consistent with the initialized solution.

(2).The three momentum equations are solved consecutively using the current value form pressure so as to update the velocity field.

(3).Since the velocities obtained in the previous step may not satisfy the continuity equation, one more equation for the pressure correction is derived from the continuity equation and the linearized momentum equations: once solved, it gives the correct pressure so that continuity is satisfied.

(4).Other equations for scalar quantities such as turbulence, chemical species and radiation are solved using the previously updated value of the other variables; when inter- phase coupling is to be considered, the source terms in the appropriate continuous phase equations have to be updated with a discrete phase trajectory calculation.

(5).Finally, the convergence of the equations set is checked and all the procedure is repeated until convergence criteria are met.

The FLUENT CFD code has extensive interactivity, so we can make changes to the analysis at any time during the process. This saves time and enables to refine designs more efficiently. Graphical user interface (GUI) is intuitive, which helps to shorten the learning curve and make the modeling process faster. In addition, FLUENT's adaptive and dynamic mesh capability is unique and

works with a wide range of physical models. This capability makes it possible and simple to model complex moving objects in relation to flow. This solver provides the broadest range of rigorous physical models that have been validated against industrial scale applications, so we can accurately simulate real-world conditions, including multiphase flows, reacting flows, rotating equipment, moving and deforming objects, turbulence, radiation, acoustics and dynamic meshing.

1.3 Solver

The CFD solver does the flow calculations and produces the results. The numerical solution of Navier–Stokes equations in CFD codes usually implies a discretization method: it means that derivatives in partial differential equations are approximated by algebraic expressions which can be alternatively obtained by means of the finite-difference or the finite-element method. Otherwise, in a way that is completely different from the previous one, the discretization equations can be derived from the integral form of the conservation equations: this approach, known as the finite volume method, is implemented in FLUENT, because of its adaptability to a wide variety of grid structures. The result is a set of algebraic equations through which mass, momentum, and energy transport are predicted at discrete points in the domain. In the freeboard model that is being described, the segregated solver has been chosen so the governing equations are solved sequentially. Because the governing equations are non-linear and coupled, several iterations of the solution loop must be performed before a converged solution is obtained and each of the iteration is carried out as follows:

(1).Fluid properties are updated in relation to the current solution; if the calculation is at the first iteration, the fluid properties are updated consistent with the initialized solution.

(2).The three momentum equations are solved consecutively using the current value form pressure so as to update the velocity field.

(3).Since the velocities obtained in the previous step may not satisfy the continuity equation, one more equation for the pressure correction is derived from the continuity equation and the linearized momentum equations: once solved, it gives the correct pressure so that continuity is satisfied.

(4).Other equations for scalar quantities such as turbulence, chemical species and radiation are solved using the previously updated value of the other variables; when inter- phase coupling is to be considered, the source terms in the appropriate continuous phase equations have to be updated with a discrete phase trajectory calculation.

(5).Finally, the convergence of the equations set is checked and all the procedure is repeated until convergence criteria are met.

The FLUENT CFD code has extensive interactivity, so we can make changes to the analysis at any time during the process. This saves time and enables to refine designs more efficiently. Graphical user interface (GUI) is intuitive, which helps to shorten the learning curve and make the modeling process faster. In addition, FLUENT's adaptive and dynamic mesh capability is unique and works with a wide range of physical models. This capability makes it possible and simple to model complex moving objects in relation to flow. This solver provides the broadest range of rigorous physical models that have been validated against industrial scale applications, so we can accurately simulate real-world conditions, including multiphase flows, reacting flows, rotating equipment, moving and deforming objects, turbulence, radiation, acoustics and dynamic meshing.

DISCRETIZATION & SOLUTION ALGORITHM

2.1 Discretization

Discretization concerns the process of transforming continuous models and equations into discrete counterparts, means derivatives in partial differential equations are approximated by algebraic expressions which can be alternatively obtained by means of the finite-difference or the finite-element method. So the, differential equations are converted into algebraic equations involving unknown values at chosen grid points which are called as discretized equations.

2.2 Grid

Meshing is the task of partitioning a spatial domain into simple geometric elements such as triangles (in 2D) or tetrahedrons (in 3D). Grid (mesh) generation is often considered as the most important and most time consuming part of CFD simulation. The solution of a flow problem (velocity, pressure, temperature etc.) is defined at nodes inside each cell. The quality of the grid plays a direct role on the quality of the analysis, regardless of the flow solver used. The accuracy of CFD solutions is governed by number of cells in the grid. In general, the larger numbers of cells better the solution accuracy. Both the accuracy of the solution & its cost in terms of necessary computer hardware & calculation time are dependent on the fineness of the grid. Efforts are underway to develop CFD codes with a (self) adaptive meshing capability. Ultimately such programs will automatically refine the grid in areas of rapid variation. Additionally, the solver will be more robust and efficient when using a well-constructed mesh. Basically, there exist three different types of grids.

2.2.1 Structured Grid

Each grid point (vertex, node) is uniquely identified by the indices i, j, k and the corresponding Cartesian coordinates $x_{i,j,k}$, $y_{i,j,k}$ and $z_{i,j,k}$, the grid cells are quadrilaterals in 2D and hexahedral in 3D. In order to resolve the boundary layers accurately, generally in 2D rectangular and in 3D prismatic or hexahedral elements are employed near solid walls. Structured grids enjoy a considerable advantage over other grid methods in that they allow the user a high degree of control. With structured grids the elements can be stretched and twisted to fit the domain because the user places control points and edges interactively, User has total freedom when positioning the mesh. So, the grid is most often flow-aligned, thereby yielding greater accuracy within the solver. Structured block flow solvers typically require the lowest amount of memory for a given mesh size and execute faster because they are optimized for the structured layout of the grid. Also, post processing of the results on a structured block grid is typically a much easier task because the logical grid planes make excellent reference points for examining the flow field and plotting the results. The major drawback of structured block grids is the time and expertise required to lay out an optimal block structure for an entire model.
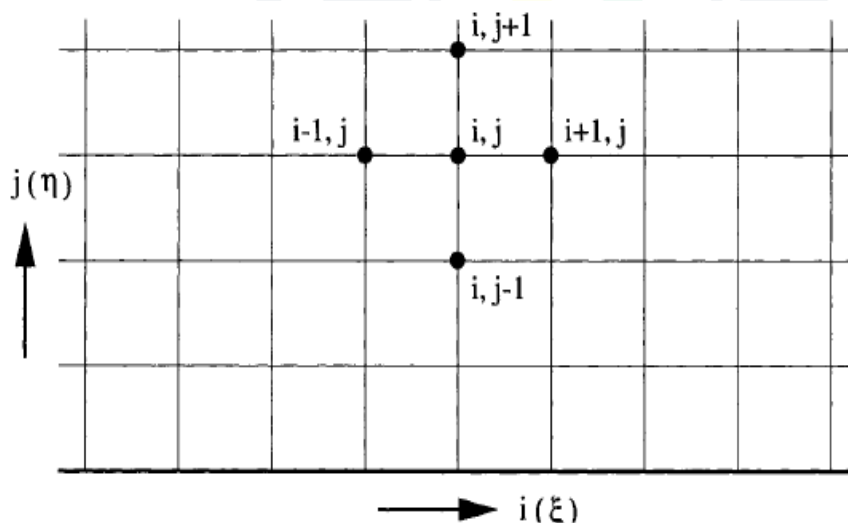

Fig. 4.1 Structured body-fitted grid approach (in 2-D), $_i$ & $_j$ represents a

Curvilinear coordinate system

2.2.2 Unstructured Grid

Grid cells as well as grid points have no particular ordering, i.e., neighbouring cells or grid points cannot be directly identified by their indices. As these grid methods utilize an arbitrary collection of elements to fill the domain. So, the mesh is called

unstructured. These types of grids typically utilize triangles in 2D and tetrahedral in 3D. The advantage of unstructured grid methods is that they are very much automated and, therefore, require little user time. The major drawback of unstructured grids is the lack of user control when laying out the mesh. Another drawback of the methods is their reliance on good CAD data. Most meshing failures are due to some error in the CAD model. Unstructured flow solvers typically require more memory and have longer execution times than structured grid solvers on a similar mesh. Post-processing the solution on an unstructured mesh requires powerful tools for interpolating the results onto planes and surfaces of rotation for easier viewing.

(a)   (b)
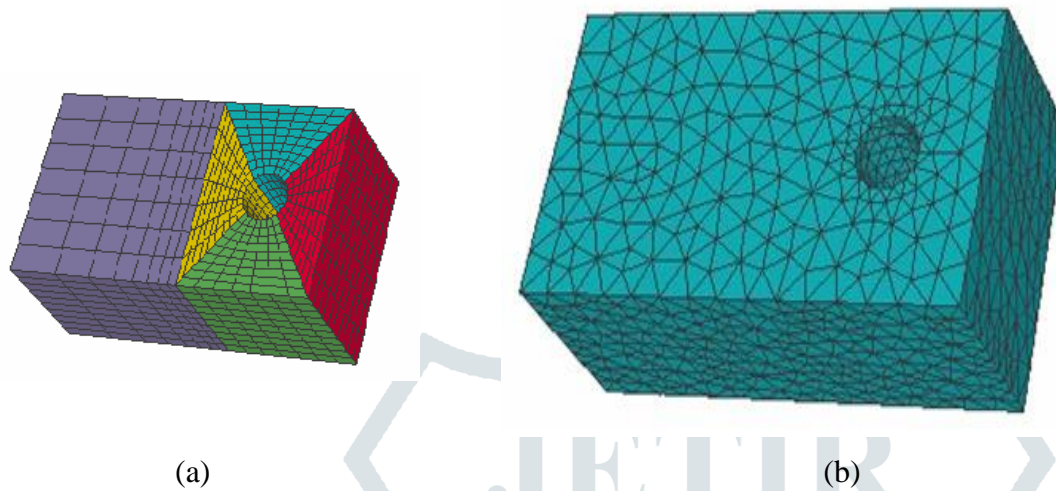
Fig. 4.2 (a) Structured grid (b) unstructured grid

IV. RESULTS AND DISCUSSION

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

1. Isotherms Pattern

Isotherms Pattern

Figure shows the instantaneous isotherm near the cylinder for various cylinder width (B=1, 2 & 3) at different Re numbers. As the width of cylinder increase there is crowding of isotherm. The maximum crowding of isotherm is seen on the front of the face, indicating the highest Nusselt number, since the thermal boundary layer growth starts from this face. As the Re number increase with increase cylinder width the recirculation region behind the cylinder generated.
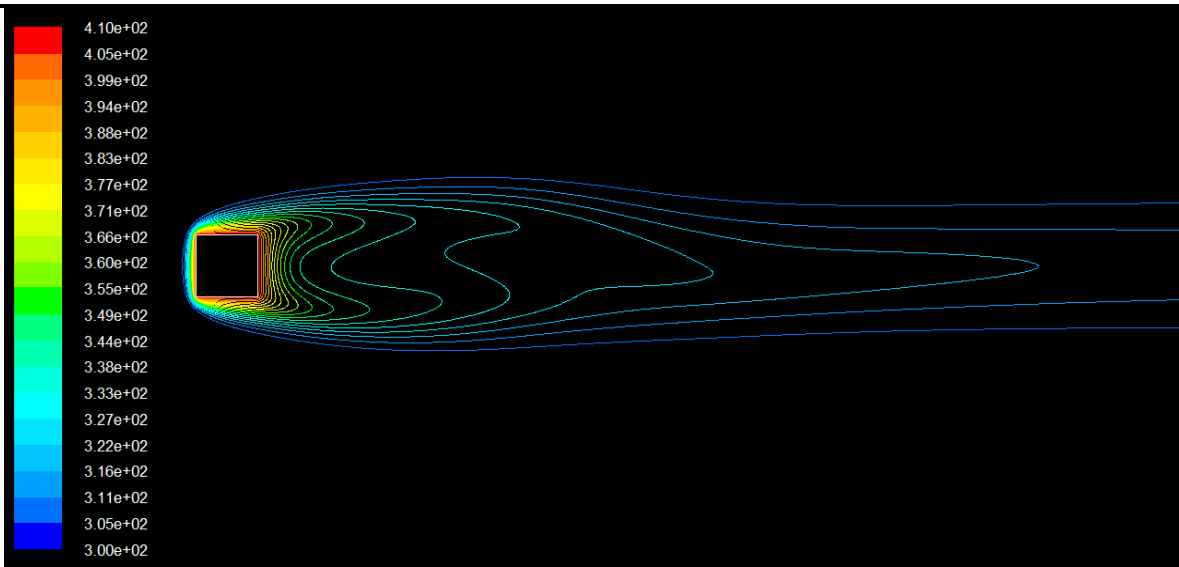
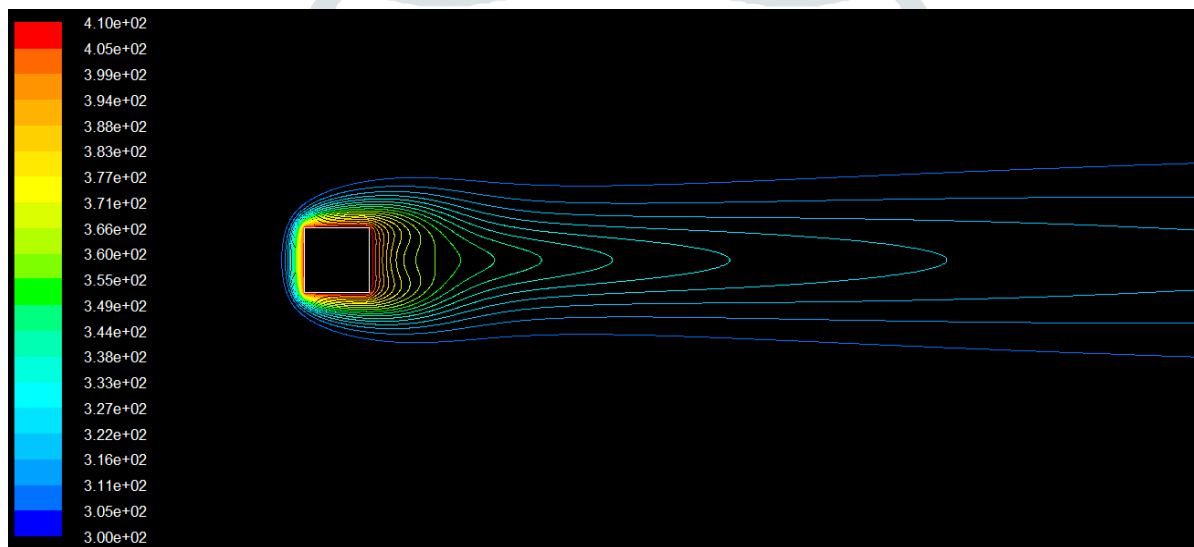Fig. 5.7 instantaneous isotherms near square cylinder (B=1, Re=100)

Fig. 5.8 instantaneous isotherms near square cylinder (B=1, Re= 40)
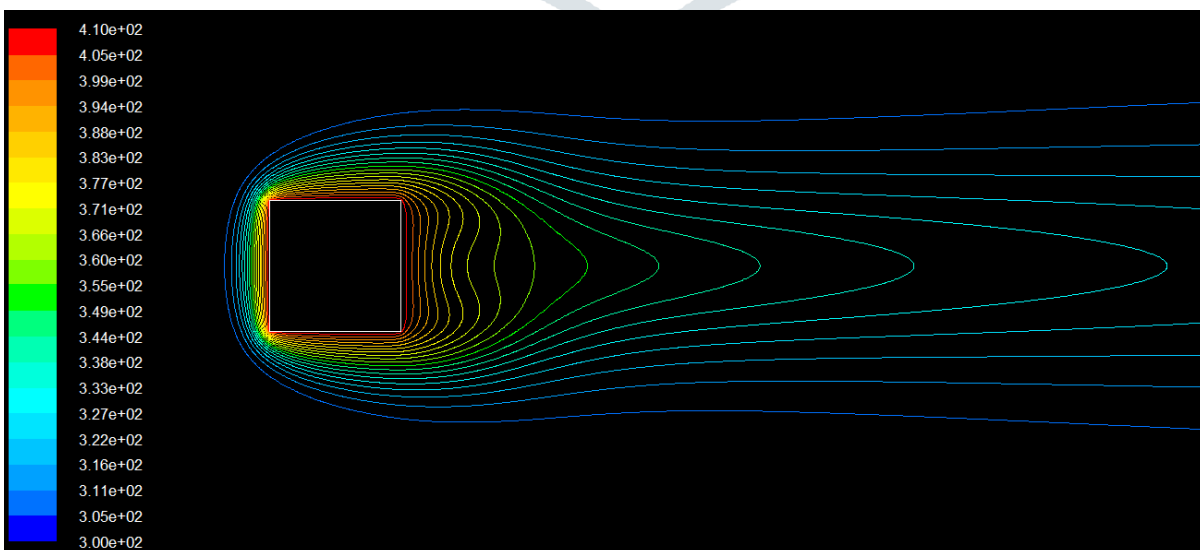

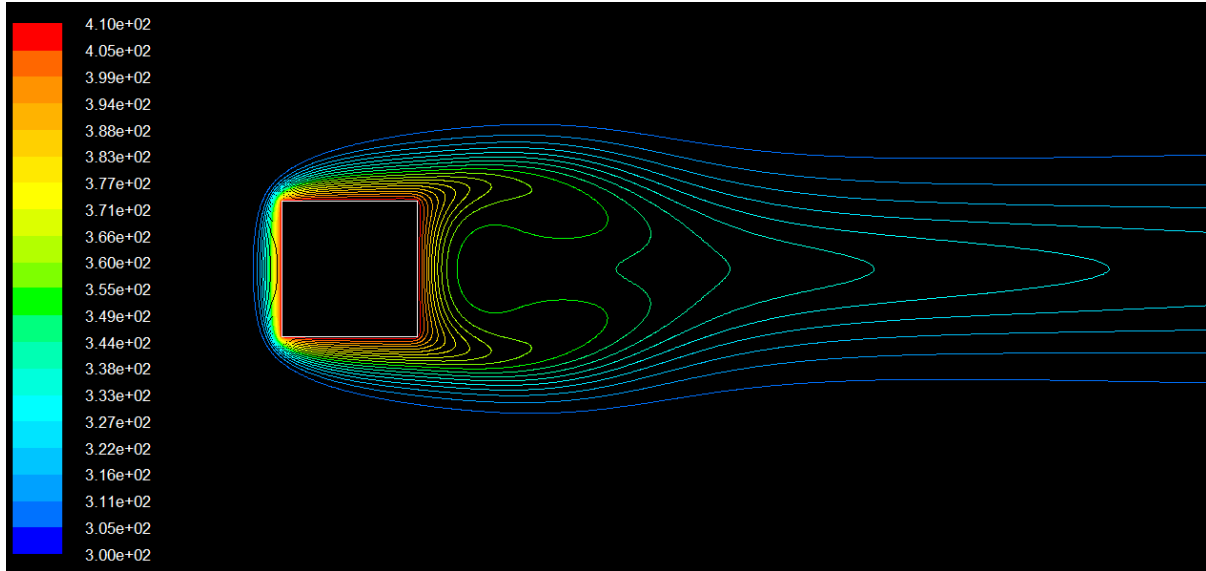Fig. 5.9 instantaneous isotherms near square cylinder (B=2, Re=20)

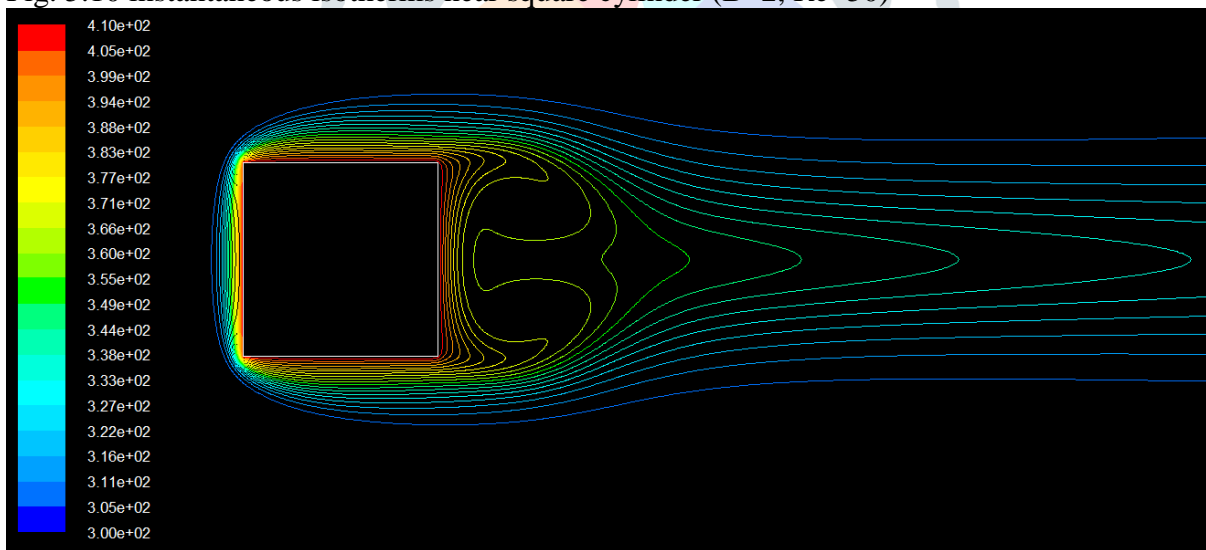Fig. 5.10 instantaneous isotherms near square cylinder (B=2, Re=50)

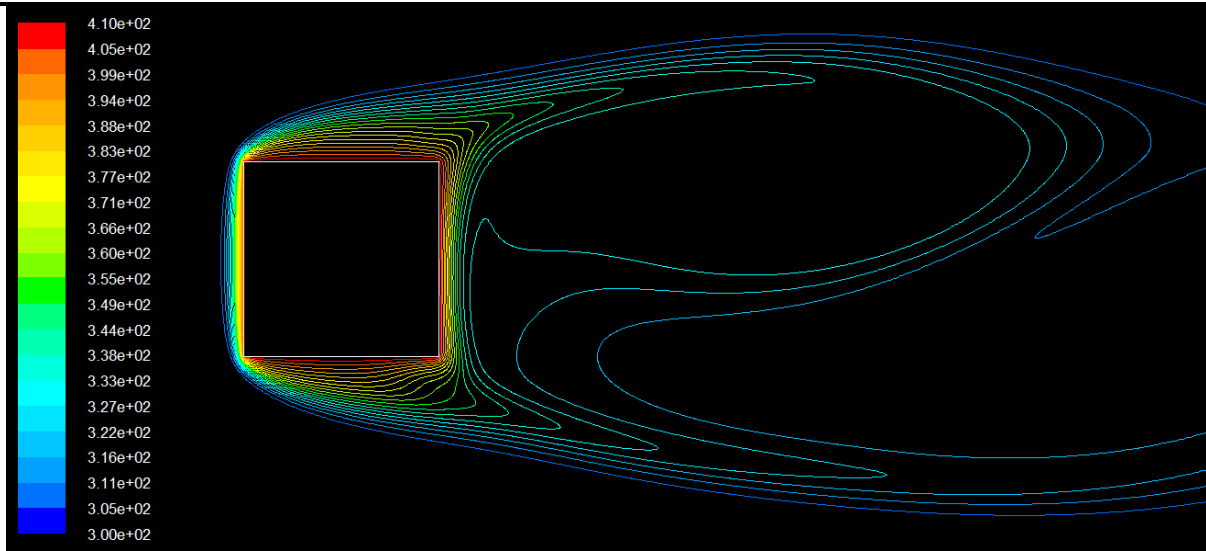Fig. 5.11 instantaneous isotherms near square cylinder (B=3, Re=50)

Fig. 5.12 instantaneous isotherms near square cylinder (B=3, Re=100)

.

REFERENCES

1. Ho, C.J., Wu, M.S. and Jou, J.B., (1990), Analysis of Buoyancy-Aided Convection Heat Transfer From a Horizontal Cylinder in Vertical Du-ct at low Reynolds Number, Warmaund Stoffudertrangung, Vol. 25 , pp. 337-343

2. Hoffmann, K.A., (1989), Computational Fluid Dynamics for Engineers, Engineering Education System, Austin.

3. Karniadakis, G.E., (1988), Numerical Simulation of Forced Convection Heat Transfer from a Cylinder in Crossflow, *Int. J. Heat Mass Transfer*, Vol. 31, pp. 107-118

4. Kelkar, K.M. and Patankar, S.V., (1992), Numerical Predication of Vortex Shedding behind a Square Cylinder, *Int. J. Numer. Methods Fluids*, Vol. 140. pp. 327-341.

5. Lange, C.F., Durst, F. and Breuer, M., (1998), Momentum and Heat Transfer from Cylinder in Laminar Crossflow at $10^{-4} \leq$ Re ≤ 200, *Int. J. Heat Mass Transfer*, Vol. 41, pp. 3409-3430

6. Mitra, N.K., Kiehm, P. and Fiebig, M., (1986), Numerical Investigation of Heat Transfer in Flows in Two-and Three-Dimensional Channels with a built-in Cylinder, AIAA Paper No.86-1329, AIAA/ASME 4th Thermophysics and Heat Transfer Conf. in Boston.

7. Morgan, V.I., (1975), The Overall Convective Heat Transfer from Smooth Circular Cylinder, Advances in Heat Transfer, Academic Press, Vol. 11, pp. 199-264.