

UDC 519.63; 519.684

* Issakhov A., Shaibekova A.A.

Faculty of Mechanic-Mathematics,
al-Farabi Kazakh National University, Almaty, Kazakhstan
*e-mail: alibek.issakhov@gmail.com

Mathematical modelling of flow around obstacles with complex geometric configuration in a viscous incompressible medium

Abstract. In this paper, we numerically investigate flow around obstacles with complex geometric configuration in a viscous incompressible environment. The Navier-Stokes equations were used to modeling the flow around obstacles with complex geometric configuration. The numerical algorithm was constructed by using projection method. At the first stage the intermediate speed is determined by the 5-step Runge-Kutta method. At the second stage the results of intermediate velocity used to determine the pressure field. Poisson equation for the pressure field is solved numerically by using the Jacobi method. The numerical algorithm is tested at flow around the square cylinder and compared with experimental data, which gives good results. Also, in this work simulated non-stationary flow around one and two cylinders obstacles arranged opposite each other.

Key words: The Navier-Stokes equations, finite volumes method, Karman vortex shedding, an aerodynamic tube.

Introduction

One of the main problems of the mechanics is the study of viscous incompressible environment. Flow around obstacles with complex geometric configuration, such as flow around a cylinder with a circular cross-section, is well-studied problem. In many mechanical engineering applications, separated flows often appear around any technical object. Tall buildings, monuments, and towers are permanently exposed to wind. Particular attention to the aerodynamic stability of constructions began to pay after some unfortunate incidents such as the collapse of a hanging bridge over the river Tacoma (USA, November 7, 1940), three towers on the thermal power plant Ferry bridge (England, November 1, 1965), the accident at the Nuclear Power Plant Turkey Point (USA, 1985). Very often, research is conducted on the examples of two-dimensional cylinder flow problems varying cross-sectional shape: circular, square, rectangular, elliptical, etc. In this paper tested numerical algorithm as an example of flow of the square cylinder and a study of non-stationary two-dimensional flow over a cylinder and two cylinders with a circular cross-section, located opposite each other.

Mathematical formulation of the problem

To describe the motion of the liquid and the gas around the geometric configuration of a viscous incompressible unsteady environment used two-dimensional Navier-Stokes equations with constant density and kinematic viscosity: In our case, the system consists of two equations: motion equations and the continuity equation.

$$\begin{cases} \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \\ \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \mathcal{G} \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \\ \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \mathcal{G} \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) \end{cases}$$

where u, v – velocity components, p – pressure, ρ – density, ν – the kinematic viscosity, t – time, x, y – space coordinates.

In this paper for testing the numerical method is considered a two-dimensional problem of flow around a square cylinder in a viscous incompressible environment. Results obtained from a numerical simulation were compared with

experimental data provided in paper [3]. Also in this work considered two problems with flow around:
 1) structure with a circular cross-sectional shape;
 2) two structures with a circular cross-sectional shape located opposite each other.

Numerical algorithm

Projection method is used to solve the Navier-Stokes equations (1) [7, 10]. It is assumed that the transfer of momentum carried out only by convection and diffusion in the first step. The intermediate velocity field is determined by the 5-step of Runge-Kutta method. In the second stage, due to the known intermediate velocity field, the pressure field is found. Poisson equation for the pressure field is solved by Jacobi method [4]. In the third step it is assumed that the transfer is carried out only by the pressure gradient [8, 9], i.e.:

$$I. \int_{\Omega} \frac{\vec{u}^* - \vec{u}^n}{\tau} d\Omega = - \oint_{\partial\Omega} \left(\nabla \vec{u}^n \vec{u}^* - \nu \Delta \vec{u}^* \right) n_i d\Gamma$$

$$II. \oint_{\partial\Omega} (\Delta p) d\Gamma = \int_{\Omega} \frac{\nabla \vec{u}}{\tau} d\Omega$$

$$III. \frac{\vec{u}^{n+1} - \vec{u}^*}{\tau} = -\nabla p$$

Experimental system of viscous flow over a square cylinder

Consider the problem of two-dimensional square-shaped cross-flow cross-section of the tubular body limitless laminar flow. [4]. A rectangular Plexiglas channel (Dimensions (m): L = 0.9255 stream wise; B = 0.01 transverse; W = 0.1 span wise) was used as the primary test section in a closed-loop configuration. A stainless steel square cylinder of diameter (D = 0.0025 m) was symmetrically affixed with the help of two miniature screws at a distance of 0.693 m from the inlet to the test section. The blockage ratio ($\beta = D/B$) was fixed at 0.25. The device D (Dimensions (m): L = 0.4; B = 0.2; W = 0.5) was connected to the test section using an intermediate channel (Dimensions (m): L = 0.5; B = 0.04; W = 0.1). This device effectively distributes the flow at the entrance to ensure a uniform and constant flow rate at the inlet of the test section. This channel had the hole (0.25 inch diameter) which was used to inject a colored dye through a streamlined stainless steel tube (1 mm diameter).

Furthermore, the apparatus 1 m length were installed inside the machine, to remove any swirl. Water flow comes from HT tank by a pump P to reservoir CHT1. CHT1 tank was connected to the apparatus through D FV valve regulating the amount of water flow. For smooth transition of the water flow from the apparatus A to the intermediate conduit used QE1, QE2 elliptical equipment. (Figure 1).

The Figure 2 illustrates a scheme of the test section that shows lines across which the time-averaged velocity profiles were measured. Physical scheme and the boundaries of computational domain are shown in Figure 1.

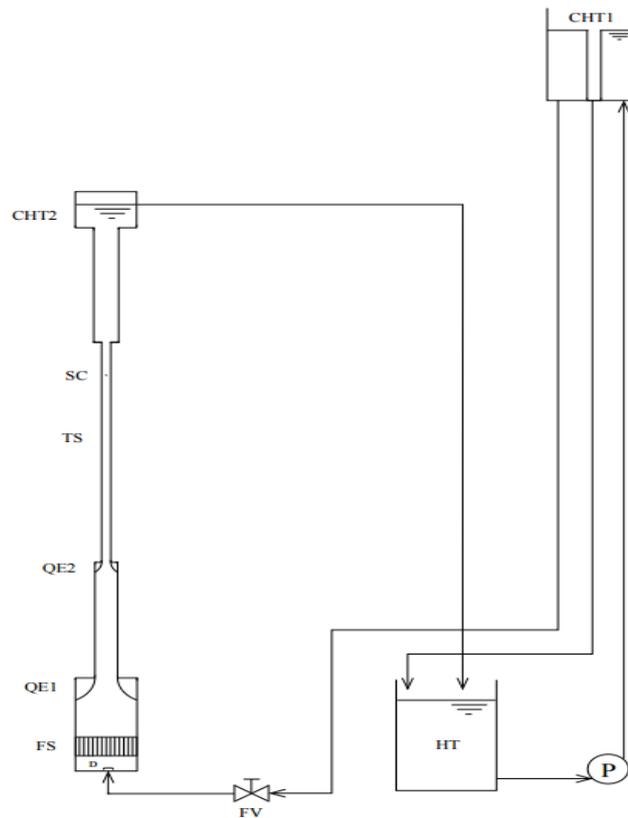


Figure 1 – Scheme of experimental system

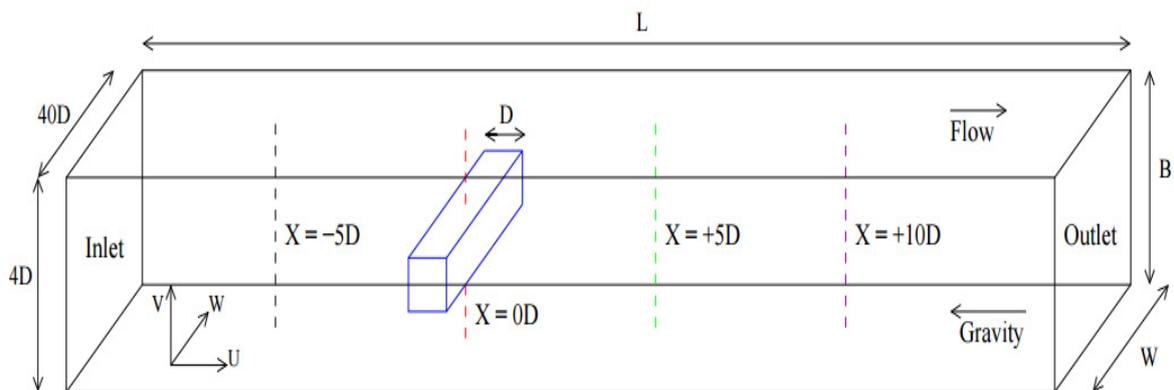


Figure 2 – Scheme of test section

Results of numerical simulation

The system of equations (1) is closed by the following boundary conditions: for the velocity components are set on the wall no slip condition and not overflow and at the inlet – parabolic profile, at the exit –

"outlet" boundary conditions. For the pressure on all borders set Neumann conditions. The parameters of the computational domain are presented in Figure 3 for the numerical simulation of flow around square cylinder problem in a viscous incompressible environment and comparing the results with the known results.

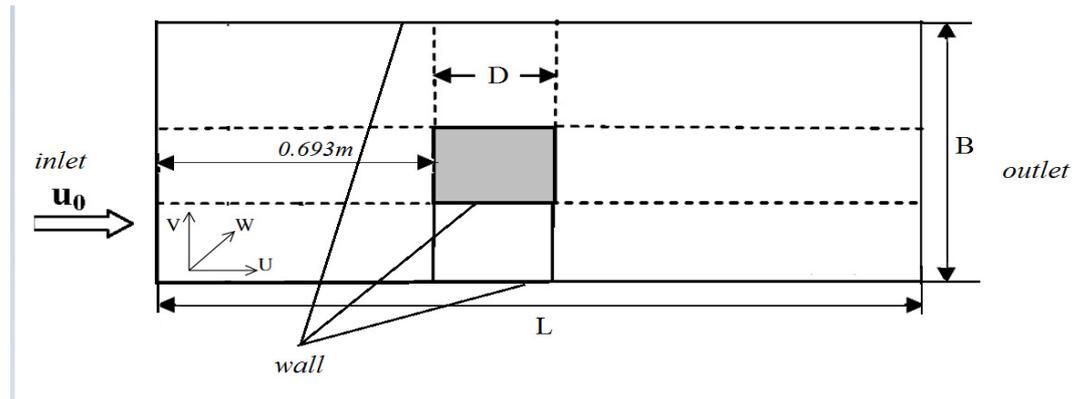


Figure 3 – Scheme of flow around a square cylinder in a viscous incompressible environment

By using a numerical algorithm to the problem of flow around a square cylinder, the following (Figure 4) was obtained:

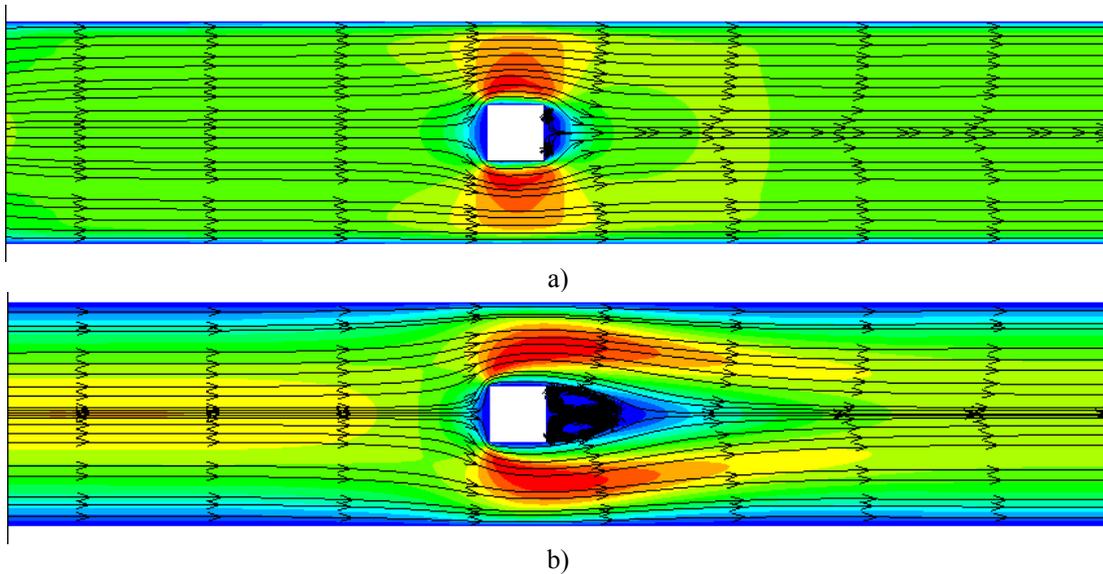


Figure 4 – The components of the velocity and streamlines the flow around a square cylinder with the number of Reynolds number $Re = 46$ for different time layers a) $t = 0.5$ sec; b) $t = 2$ sec; a) $t = 3.5$ sec

The numerical study of flow around square cylinder viscous flow charts velocity profiles were obtained and compared with the numerical and experimental results, given in paper [3]. (Figure 5) From Figure 5 it can be seen that this problem from the numerical study results are in good agreement with the experimental data [3].

With the help of the developed program of calculations to determine the velocity field over time, acting on a streamlined cylinder circular shape in a viscous incompressible environment including at $Re = 1000$ on a uniform mesh size of 225×500

was performed. In the case of low Reynolds number, the smaller the critical Reynolds number Re^* is formed within the stationary unseparated. According to published papers, the critical Reynolds number for flow past a circular cylinder is equal to $Re^*(40-50)$ [5].

Figure 6 indicates that at $Re = 1000$ is one of the vortices of the circular cylinder becomes so long that it breaks off and floats downstream with the fluid. The liquid for the cylinder is twisted again and formed new vortices. This process is known as Karman vortex shedding flow [5] (Figure 6).

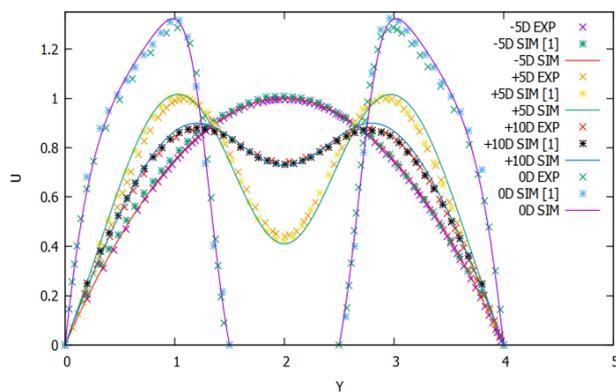


Figure 5 – Velocity profiles in certain parts of the computational domain at $Re = 46$

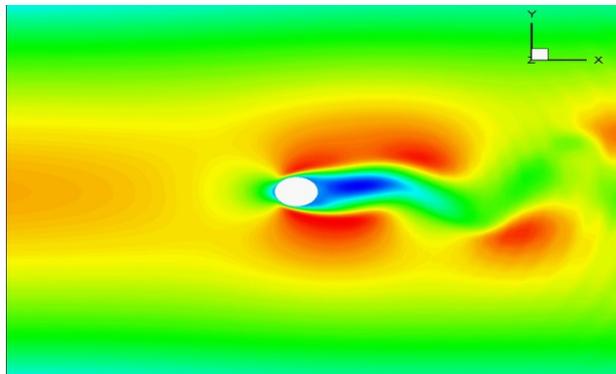


Figure 6 – Velocity contours at a Reynolds number of 1000 and $t = 2$ sec.

Flow around two circular cylinders located opposite each other was examined at $Re = 1000$ distance between the cylinders $L = 1$, the height of the computational domain $D = 2.2$. Depending on the distance between the two cylinders are formed different types of flows [5].

Figure 7 shows the flow rates at various time points when the distance between the cylinders is $L = 2.2$. With these arrangement of cylinders formed two vortices that behave more independently. (Figure 7).

By reducing the distance between the cylinders, it can be seen that the vortices formed near the cylinders influence each other [6].

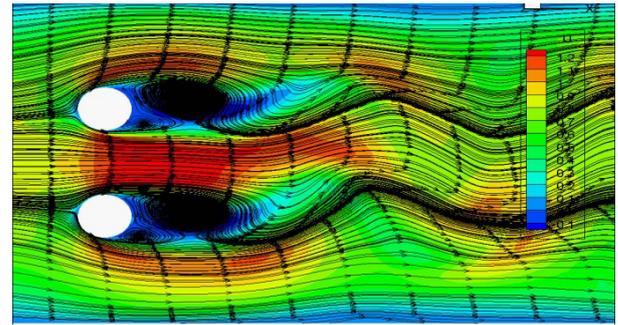


Figure 7 – Velocity contours at a Reynolds number of 1000 and $t = 2$ sec.

Conclusion

In carrying out work on the basis of numerical solution of two-dimensional Navier-Stokes equations conducted test calculations of the method used for the problem of flow around a square cylinder in a viscous medium, and studied flow around a circular cylinder.

As a result of the study, the data obtained can be used during the installation and location of the building constructions, taking into account the aerodynamic elasticity.

It should be noted that the advantages discussed in the approach are: simplicity of software implementation schemes of high order of spatial sampling, the effectiveness of its application for the calculation of the steady and unsteady flows.

References

1. Chung T.J. Computational fluid dynamics //Cambridge University Press. – 2002. – 1012 p.
2. Fletcher K. Computational Methods in Fluid Dynamics (Russian) – M., 1991. –552 p.
3. Madhavan S. Transition to three-dimensional models for flow past a confined square cylinder. PhD thesis, University of Alberta, Edmonton, AB. – Canada, 2011.
4. Anderson D., Tannehill John., Pletcher R. Computational fluid mechanics and heat transfer (Russian). – M., 1990. – 337 p.
5. Kozlov I.M., Dobergo K.V., Gnesdilov N.N. Application Of RES methods for computation of

hydrodynamic flows by an example of 2D flow past a circular cylinder for $Re=5-200$ // *International Journal of Heat and Mass Transfer*. – 2011. – Vol. 54. – P. 887-893.

6. Belov I. *Interaction of the Non-Uniform Flows with the Obstacles*. – M.: Engineering, 1983. – 166 p.

7. Issakhov A., Mathematical modeling of the discharged heat water effect on the aquatic environment from thermal power plant // *International Journal of Nonlinear Science and Numerical Simulation*. – 2015. – Vol. 16. – No 5. – P. 229–238, doi:10.1515/ijnsns-2015-0047.

8. Issakhov A., Mathematical modeling of the discharged heat water effect on the aquatic environment from thermal power plant under various operational capacities // *Applied Mathematical Modelling*. – 2016. – Vol. 40. – No 2. – P. 1082–1096

9. Issakhov A. Large eddy simulation of turbulent mixing by using 3D decomposition method // *J. Phys.: Conf. Ser.* – 2011. – Vol. 318, No 4. – P. 1282-1288, doi:10.1088/1742-6596/318/4/042051.

10. Chorin A.J. Numerical solution of the Navier-Stokes equations // *Math. Comp.* – 1968. – Vol. 22. – P. 745-762.