DIRECT NUMERICAL SIMULATION OF HIGH REYNOLDS NUMBER FLOWS BY USING THIRD ORDER UPWIND SCHEME

H. N. Mondal and S. H. Khan
Department of Chemical Engineering
Bangladesh University of Engineering and Technology

Abstract:

The present work aims to use direct numerical simulations and their mathematical analysis as a tool for turbulence research. Its long term goal concentrates around enlarging the understanding of turbulence and transition in physical terms such as energy transport, mixing, combustion etc. and in mathematical terms such as bifurcation scenarios, proper orthogonal decomposition, attractor structure etc. Ultimately it should lead to improved turbulence models for industrial simulation methods based on LES or Reynolds-averaged Navier-Stokes. In this work some high Reynolds number flows are simulated without using any turbulence model

Introduction

It is very difficult to solve Navier-Stokes equations at high Reynolds number due to numerical instability. This difficulty is due to the very small viscous diffusion. The eddy viscosity model introduces a rather large diffusion into the system, which stabilizes the computation. It is natural to investigate whether high Reynolds number flows can be solved without introducing a turbulence model or sub-grid modeling.

Another way to overcome the numerical instability in high-Reynolds-number flow computation is to use an upwind scheme. The stability of the first order upwind scheme is quite good, but it has a strong diffusive effect similar to the effect of molecular viscosity. Thus, it is not suitable for our purpose. The second order upwind scheme is better in this sense, but it is more unstable and cause undesirable propagation of errors.

Recently, a third order upwind scheme has been developed. Using this scheme, the flow around a circular cylinder in the critical regime was successfully simulated without using any turbulence model[1]. These results suggest that direct simulation without any turbulence model is possible if the computational scheme is stable and if its numerical diffusion does not conceal the effect of viscosity.

Although at high Reynolds numbers, the grid spacing needed to resolve the small eddies in the boundary layer is too fine for the computers available at the present time, the above computation using a relatively course grid has captured a global qualitative structure of turbulent flow. In addition, computations by Large Eddy Simulation (LES) have shown that the grid spacing need not be as fine as the theoretical assumption requires[2,3]. This means that the small-scale structure may not have much influence on the large-scale structure.

These results suggest that if we are interested only in the large-scale structure of turbulence, its direct numerical simulation is possible. Since the flow characteristics of engineering interest are usually determined by the large scale structures, direct simulation of high-Reynolds-number flow may give interesting results.

In this work, various high-Reynolds-number flows have been simulated without using any turbulence model. Contrary to other schemes, the present method does not require any special assumption. Simulation results appear to capture the Reynolds number dependence, and permits us to follow the transition to turbulence.

Governing Equations and Solution Scheme

The steady compressible full Navier-Stokes equations (NSE) are solved directly without any turbulence model using pseudo-transient algorithm. All special derivatives except those of non-linear terms are approximated by central difference for incompressible flow. The non-linear terms are approximated by a third order upwind scheme[1]:

$$\left(u\frac{\partial u}{\partial x}\right)_i = u_i\left(u_{i+2}-2u_{i+1}+9u_i-10u_{i-1}+2u_{i-2}\right)/6h\ ;\ u_i>0$$

$$= u_i\left(-2u_{i+2}+10u_{i+1}-9u_i+2u_{i-1}-u_{i-2}\right)/6h\ ;\ u_i<0,$$

This scheme has a numerical diffusion approximately expressed by fourth order derivative.

Numerical Simulation

For numerical simulation, three different cases are considered: flow around a blunt body, flow around a circular obstacle and flow over an vertical rectangular obstacle. The fluid considered is air and its physical properties are assumed constant and equal to the following values: density, 1.206 kg/m^3 and molecular viscosity, 1.80x10^{-5} Pa s. Computations are performed using MATLAB/FEMLAB. Geometry, stream function and vorticity contours are presented for different Reynolds number.

Flow Around a Blunt Body

The geometry considered is shown in Fig.-1. The Reynolds numbers, based on the projected diameter of the blunt object, considered are 10^4, $5x10^4$ and $5x10^5$. The corresponding inlet velocities were 0.1 m/s, 0.49 m/s and 4.87 m/s respectively. The computational domain is discretized with 7644 quadrilateral cells consisting of 7951 nodes. Figure-4 shows the calculated stream function and vorticity contours at different Reynolds numbers.

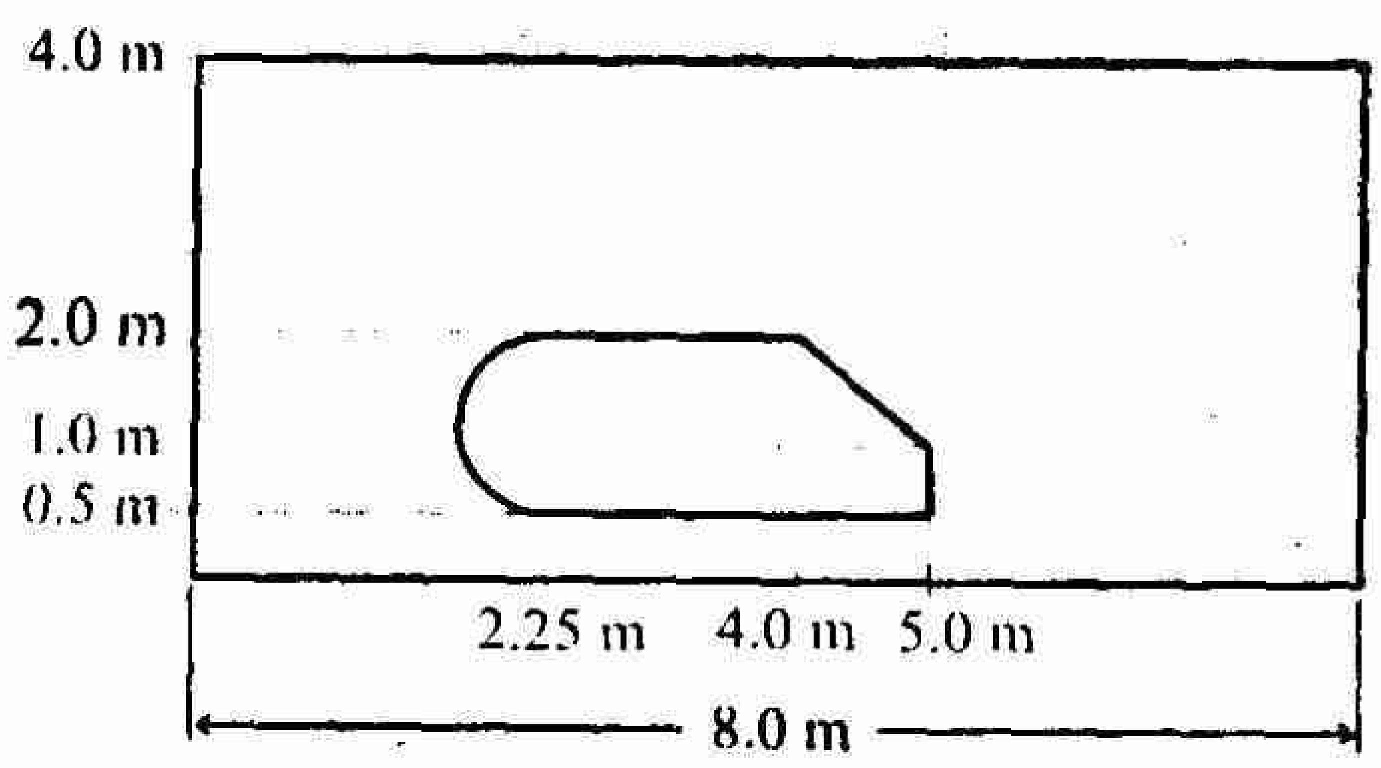


Fig. 1: Blunt object geometry

Flow Around a Circular Obstacle

The geometry of the problem is shown in Fig.-2. Calculations are performed for Reynolds numbers $2x10^4$, $6x10^4$ and $6x10^5$. The Reynolds numbers were based on the diameter of the circular obstacle. The corresponding inlet velocities considered were 4.87 m/s, 14.6 m/s and 146.07 m/s respectively. The geometry consists of 11055 quadrilateral cells with 11372 nodes. Very low under-relaxation factor is used for both velocities and pressure corrections equations to achieve converged solution. Figure-5 shows the calculated stream function and vorticity contours at different Reynolds number.

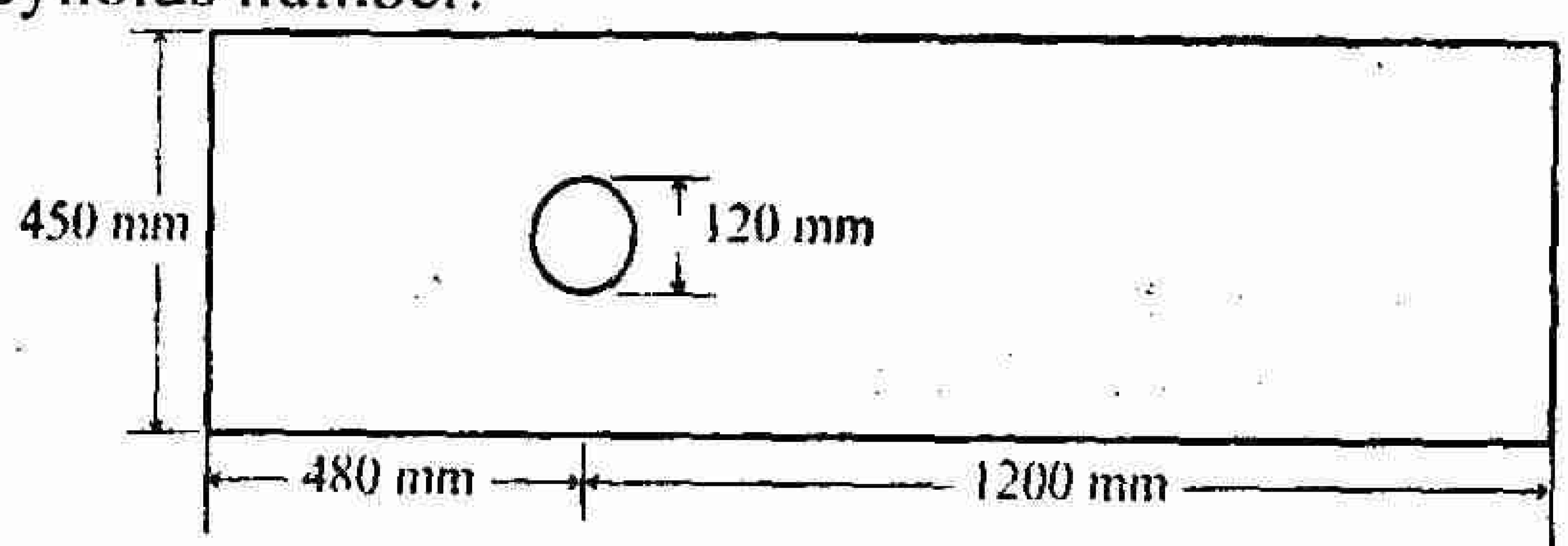


Fig. 2: Geomerty for flow around a circular object

Flow Over a Rectangular Obstacle

The problem considered is a two-dimensional flow over an obstacle mounted at the bottom wall of a two dimensional channel. The geometry is shown in Fig.-3. The computational domain is discretized with 6096 quadrilateral cells corresponding to 6315 nodes. Reynolds numbers, based on the height of the rectangular obstacle, considered are $2x10^4$, $6x10^4$ and $6x10^5$. Figure-6 shows the calculated stream function and vorticity contours at different Reynolds number.

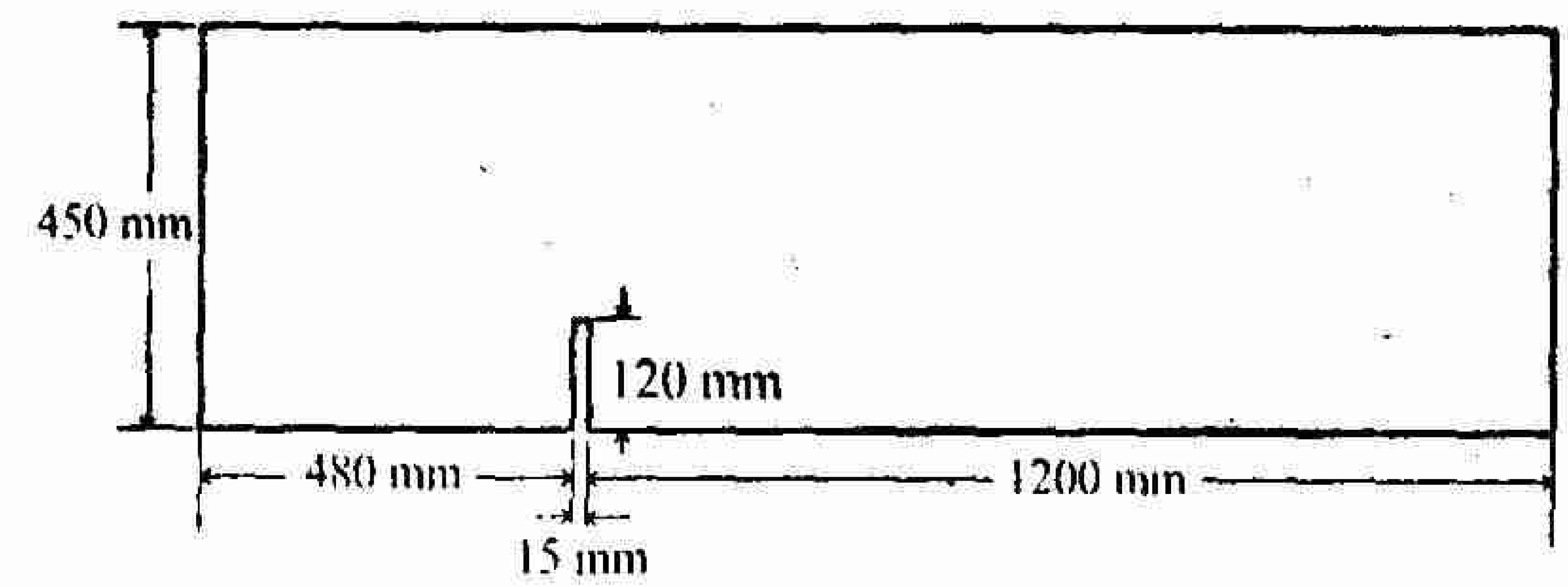


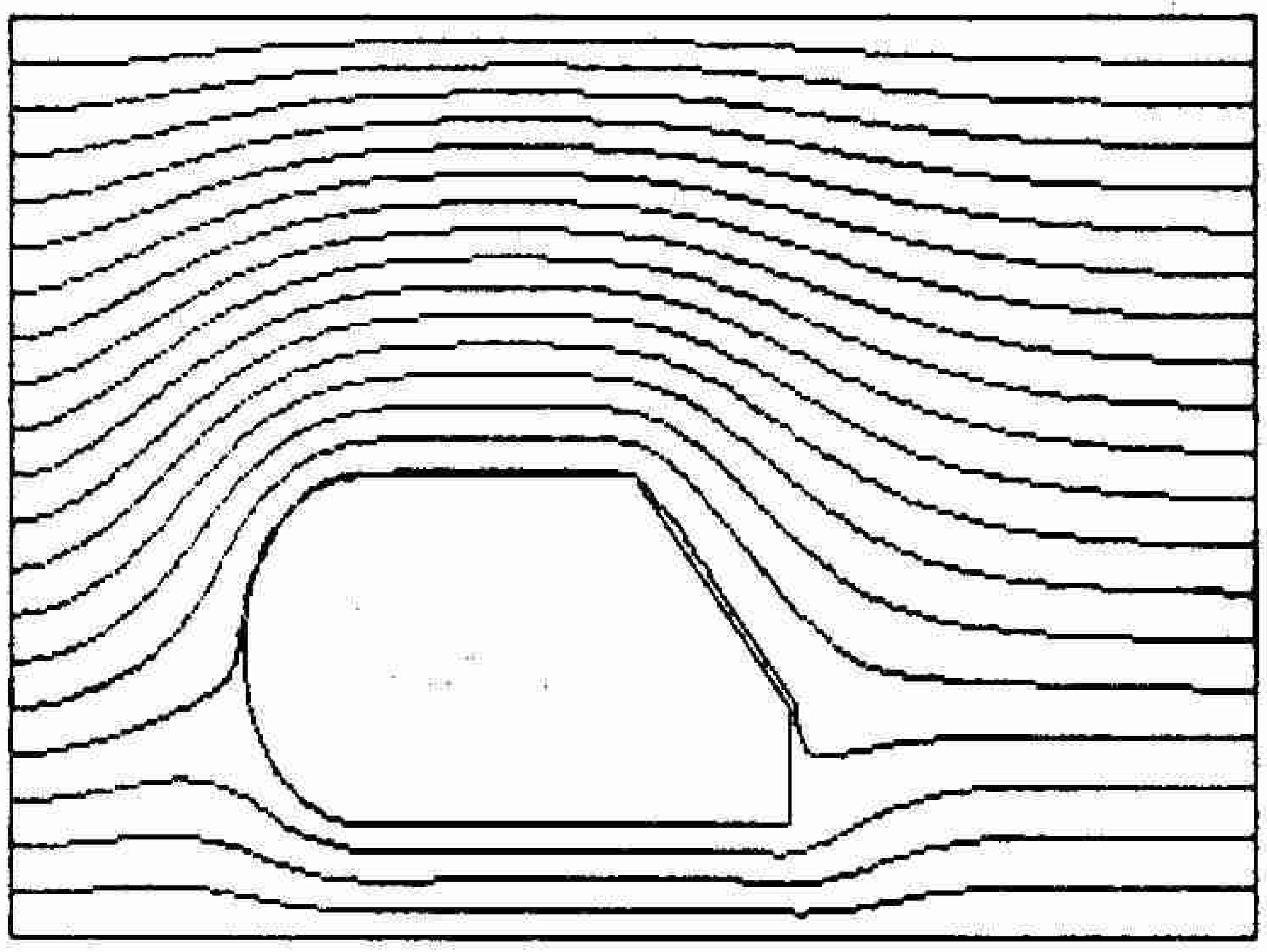
Fig. 3: Geometry for flow over a vertical obstacle

Conclusions

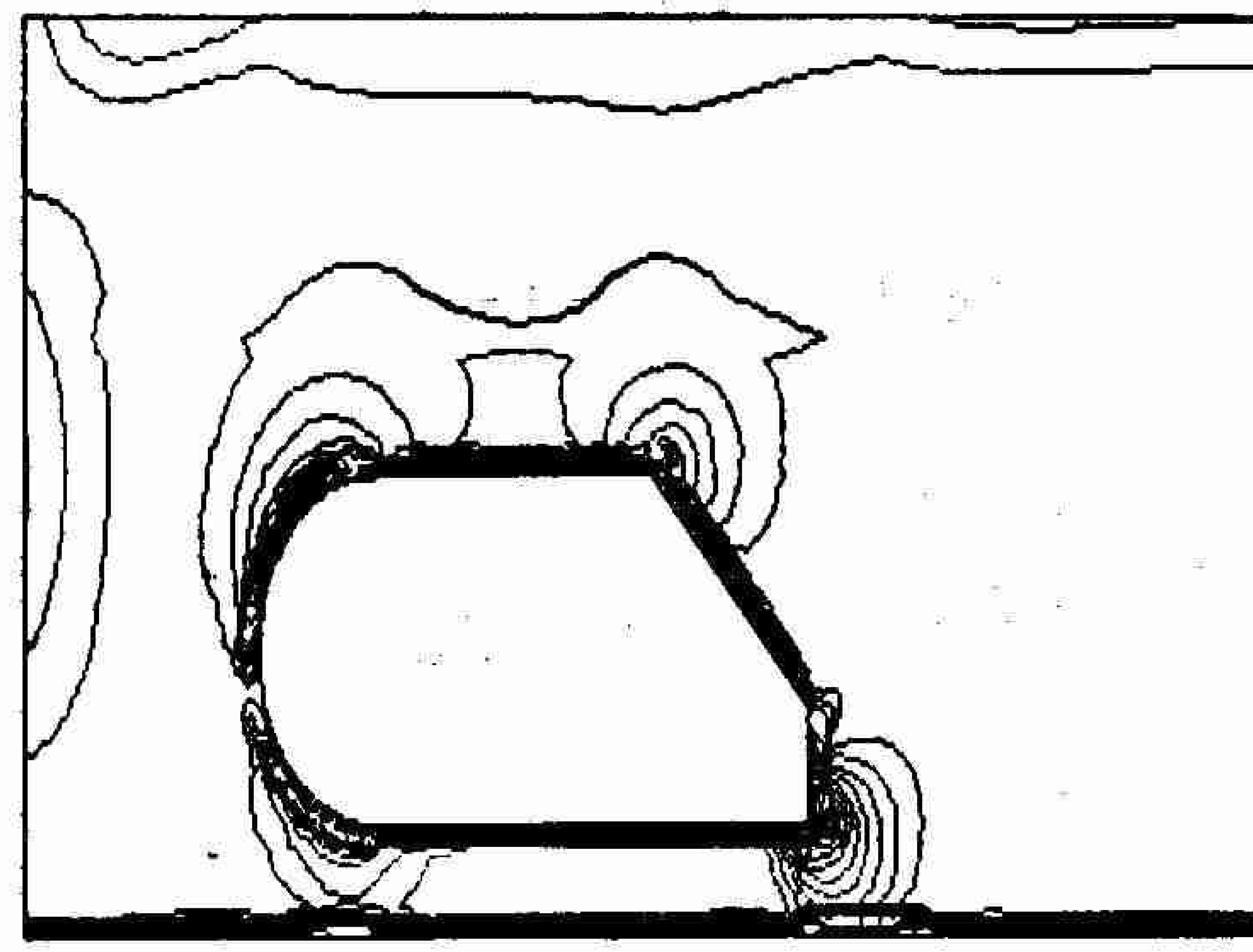
It has been shown that the structure of turbulent flows can be simulated by directly integrating the Navier-Stoks equations. The second order numerical diffusion which appears in the first order upwind scheme is similar to molecular diffusion and hides the dependence of the flow on the Reynolds number and therefore is not suitable for high Reynolds number flow calculation. On the other hand, the fourth order type numerical diffusion is of short range and does not hide the effect of molecular diffusion and at the same time stabilizes the computation very well. From theses computations, it can be concluded that high Reynolds number flows can be calculated without turbulence model at relatively high Reynolds number.

References

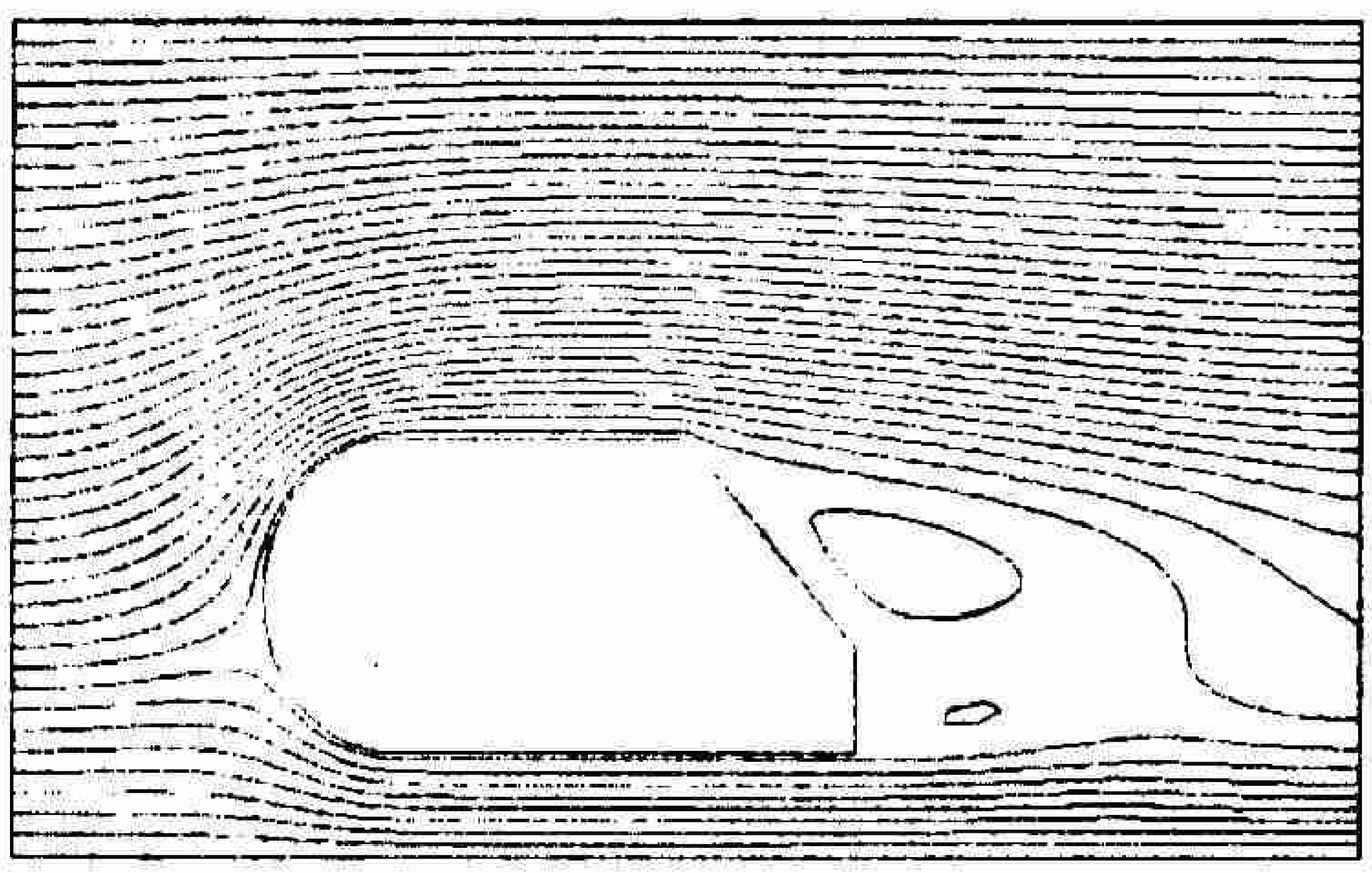
1. Kawamura, T. and Kuwahara, K., "Computation of High Reynolds Number Flow Arround a Circular Cylinder with Surface Roughness", AIAA paper 84-0340 (1984).
2. Deardorff, J.W., "A numerical study of three dimensional turbulent channel flow at large Reynolds numbers", J. Fluid Mech. 41 (1970) 453-480.
3. Kuwahara, K., "Study of flow past a circular cylinder by an invicid model", J. Phys. Soc. Jpn., 45, 292-297.
4. Stalio, E; Nobile, E; Sousa, ACM; Direct Numerical Simulation of Heat Transfer over Riblets, Proc. of IDDME 2000 and Forum 2000 SCGM/CSME Eds. Mascle, C; Fortin, C; Pegna, J; Presses Internationales Polytechnique, Paper MA22.1, Montreal, Canada, 2000.



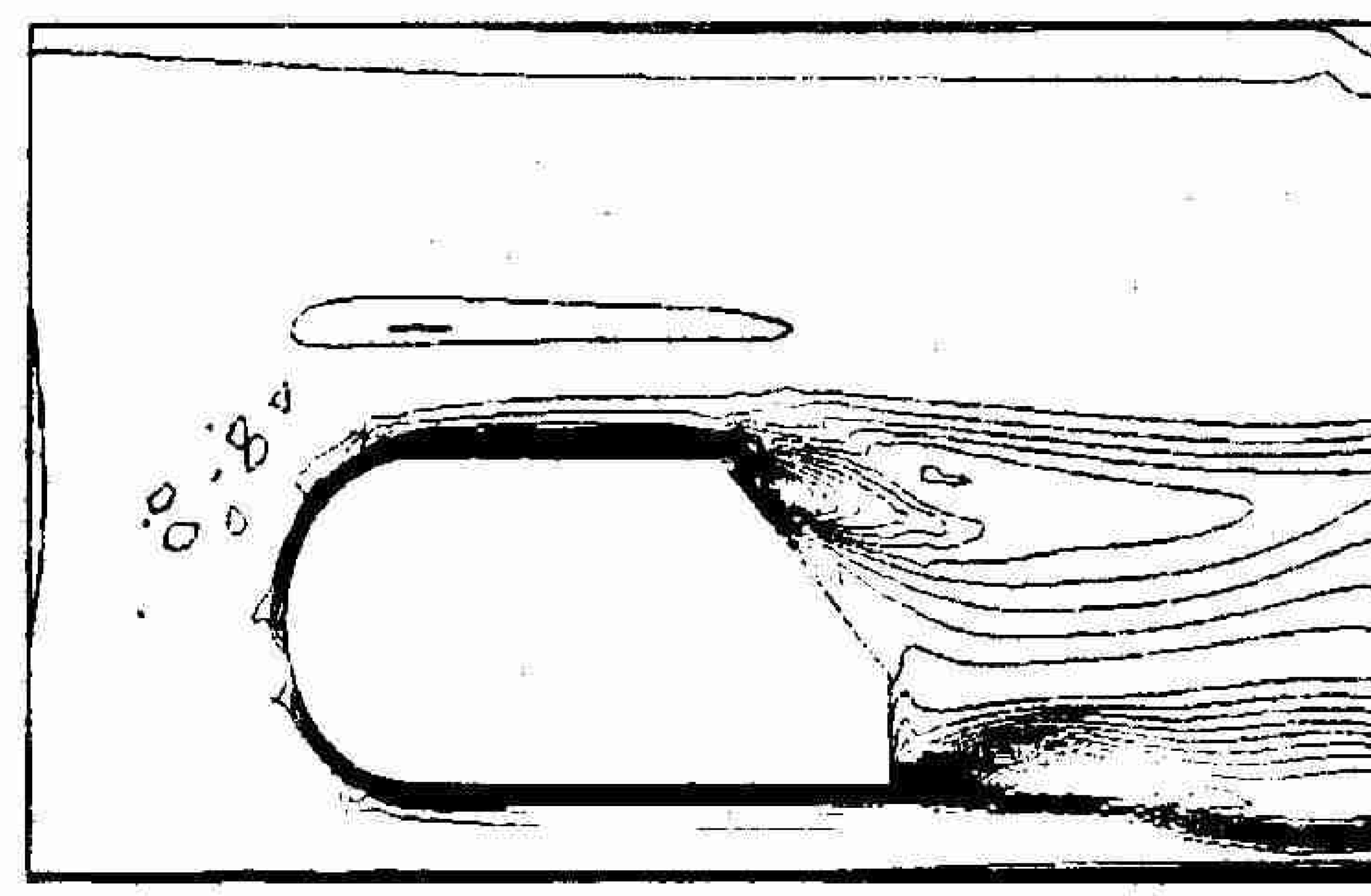
(a) Stream functions for Re=10,000



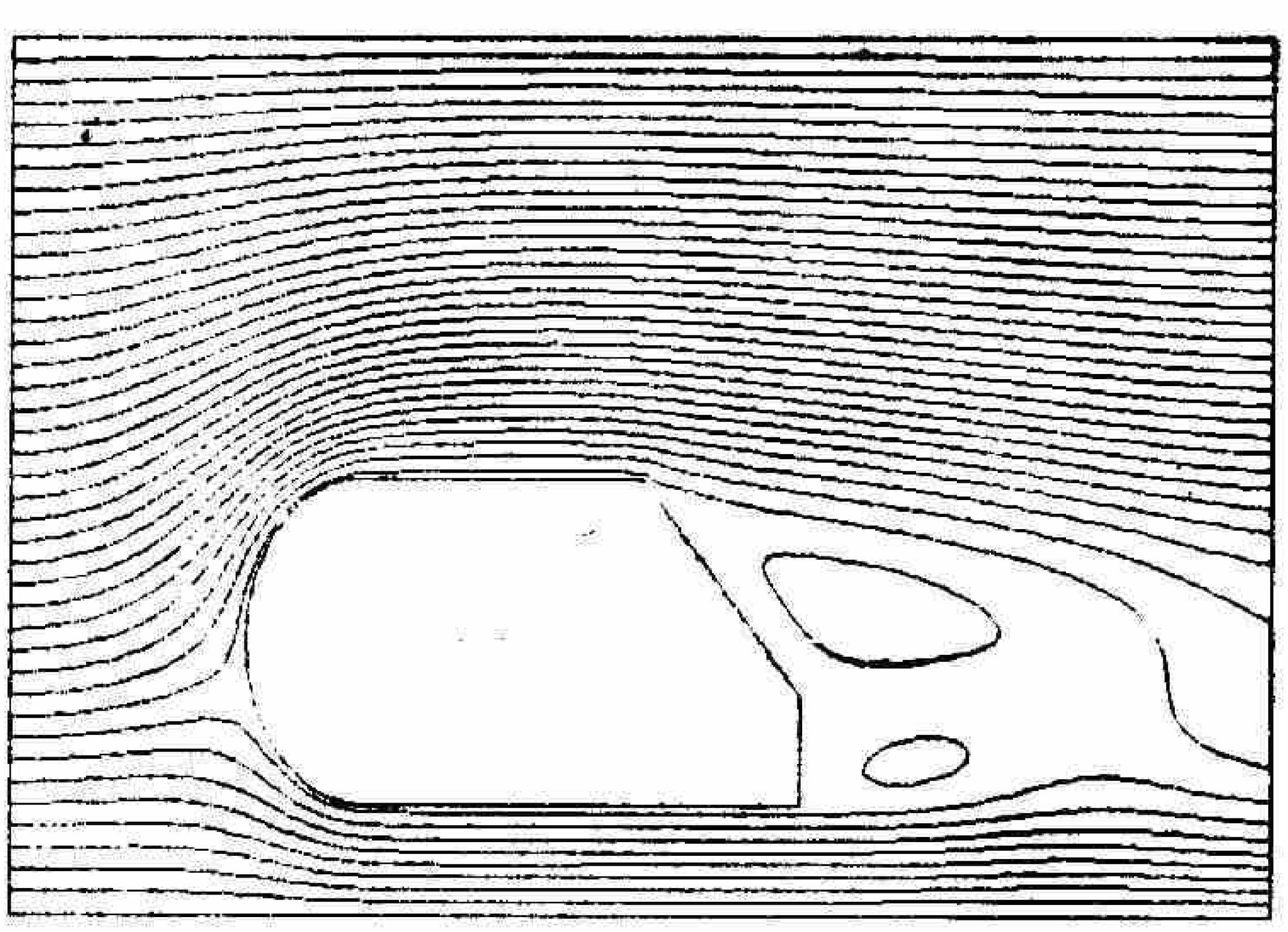
(a) Vorticity contours for Re=10,000



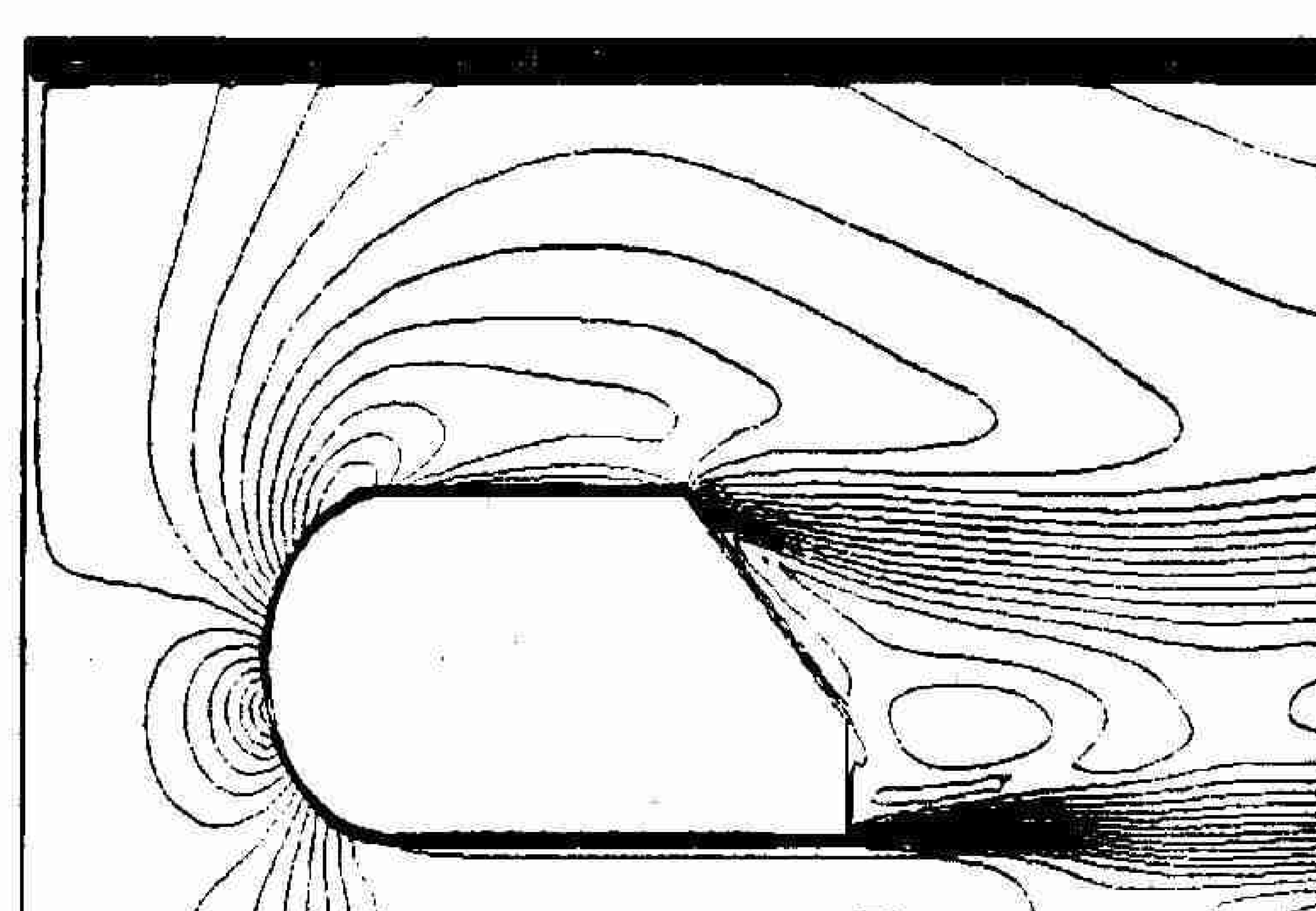
(b) Stream functions for Re=50,000



(b) Vorticity contours for Re=50,000

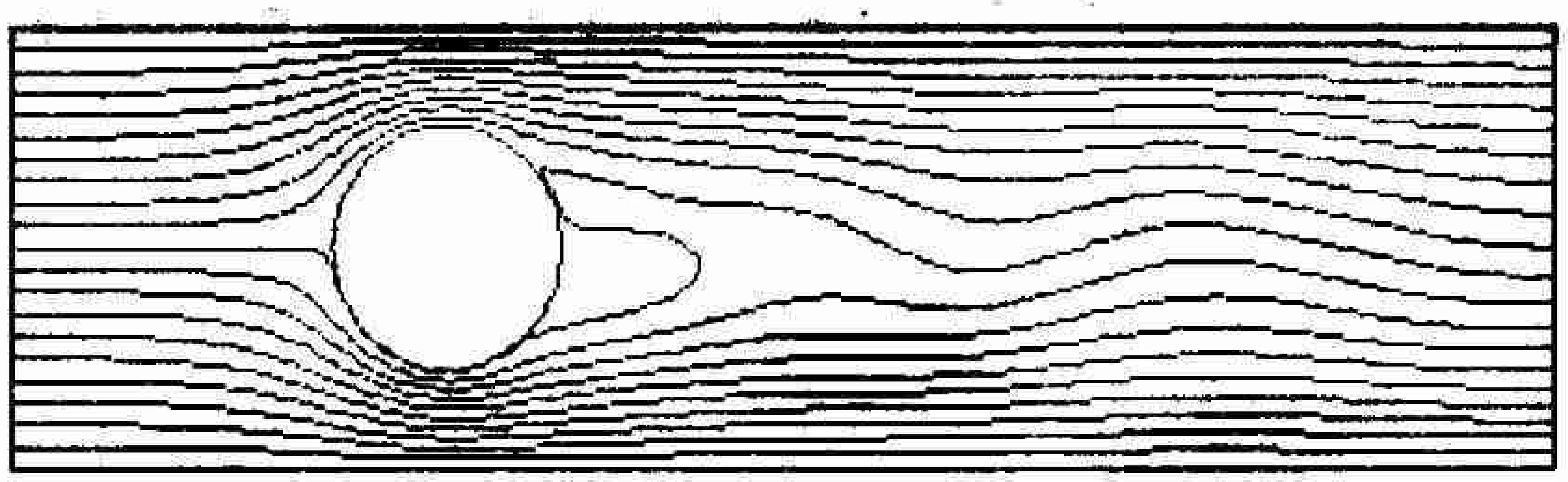


(c) Stream functions for Re=500,000

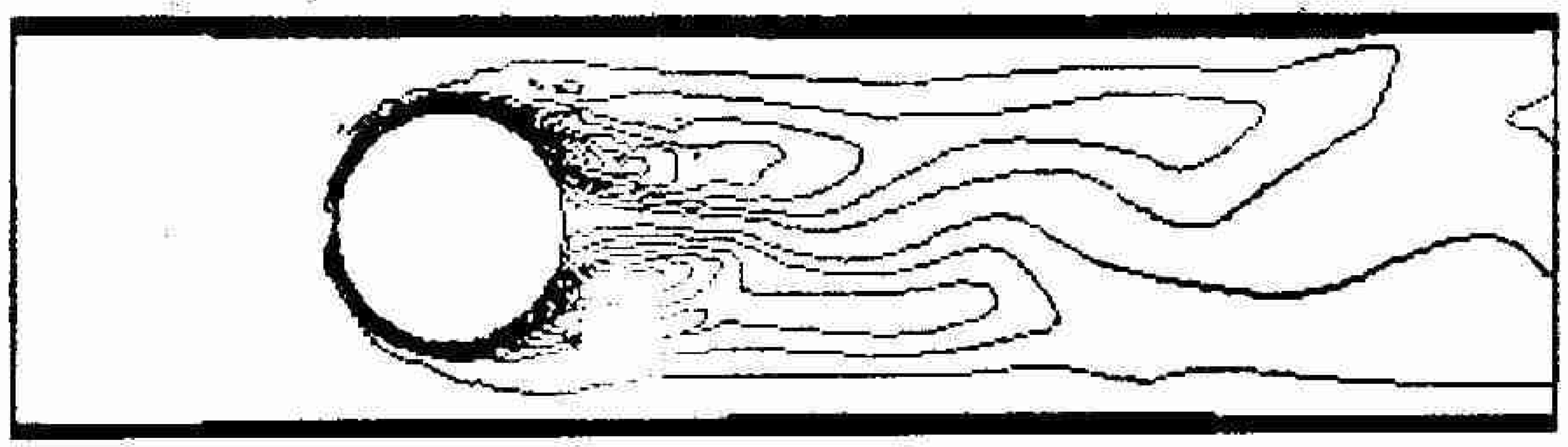


(c) Vorticity contours for Re=500,000

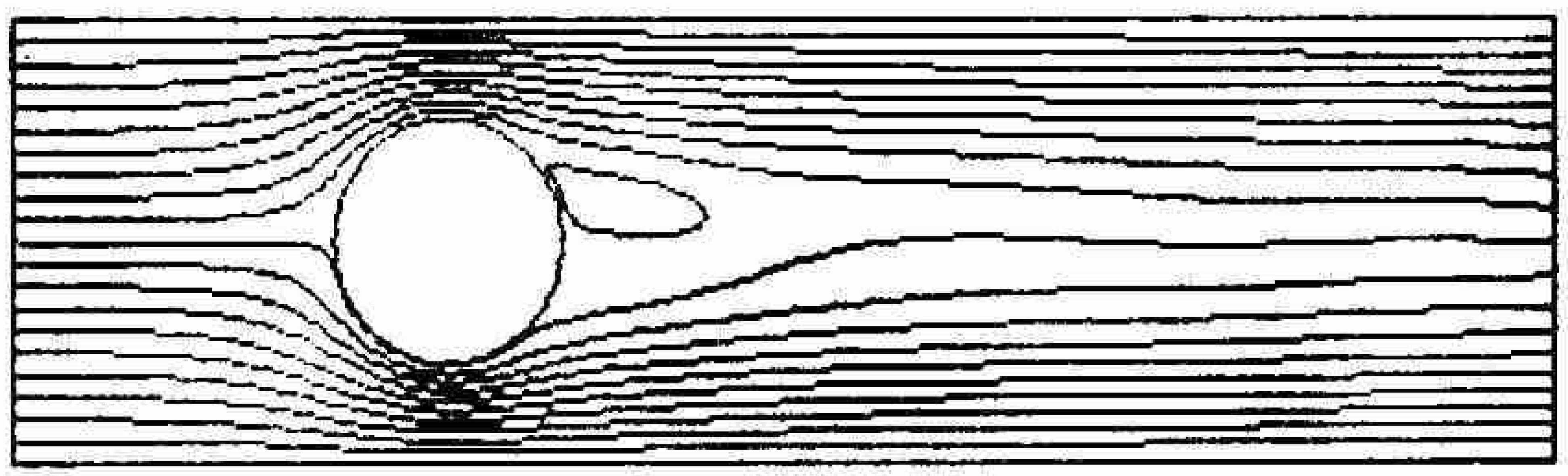
Fig. 4: Stream functions and vorticity contours for flow around a blunt body for different Reynolds numbers.



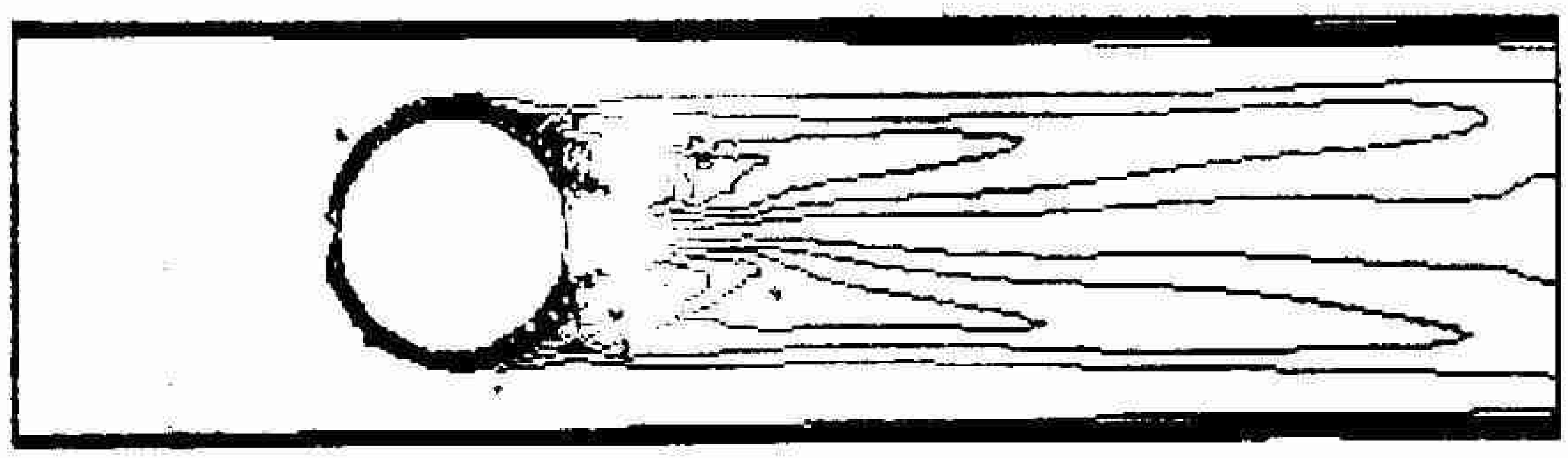
(a) Stream functions for Re=20,000



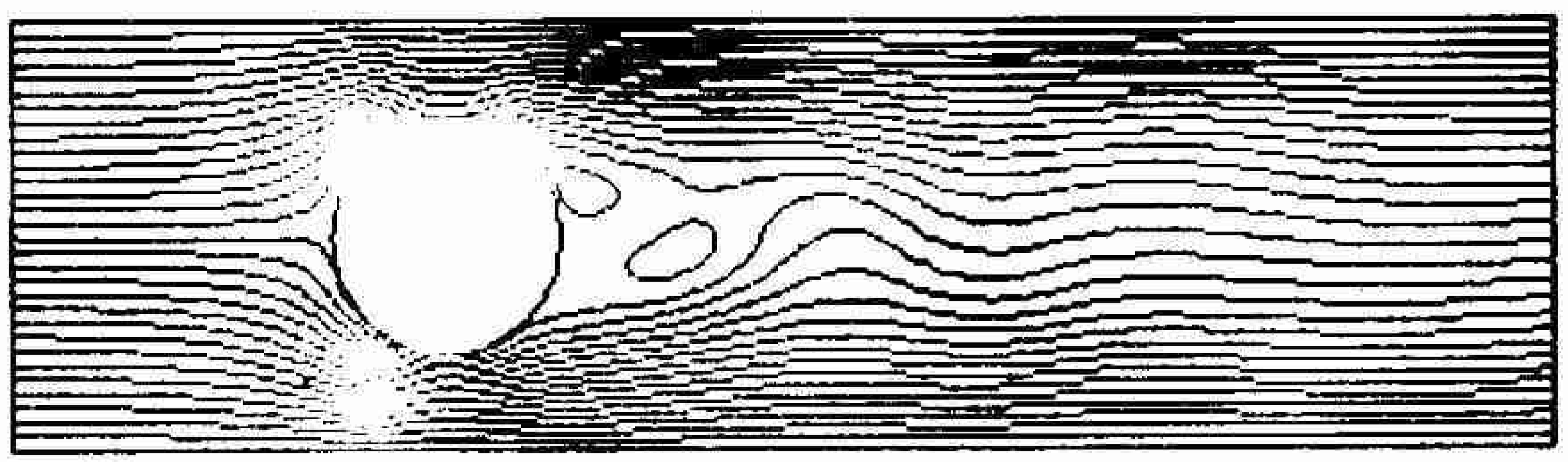
(a) Vorticity contour for Re=20,000



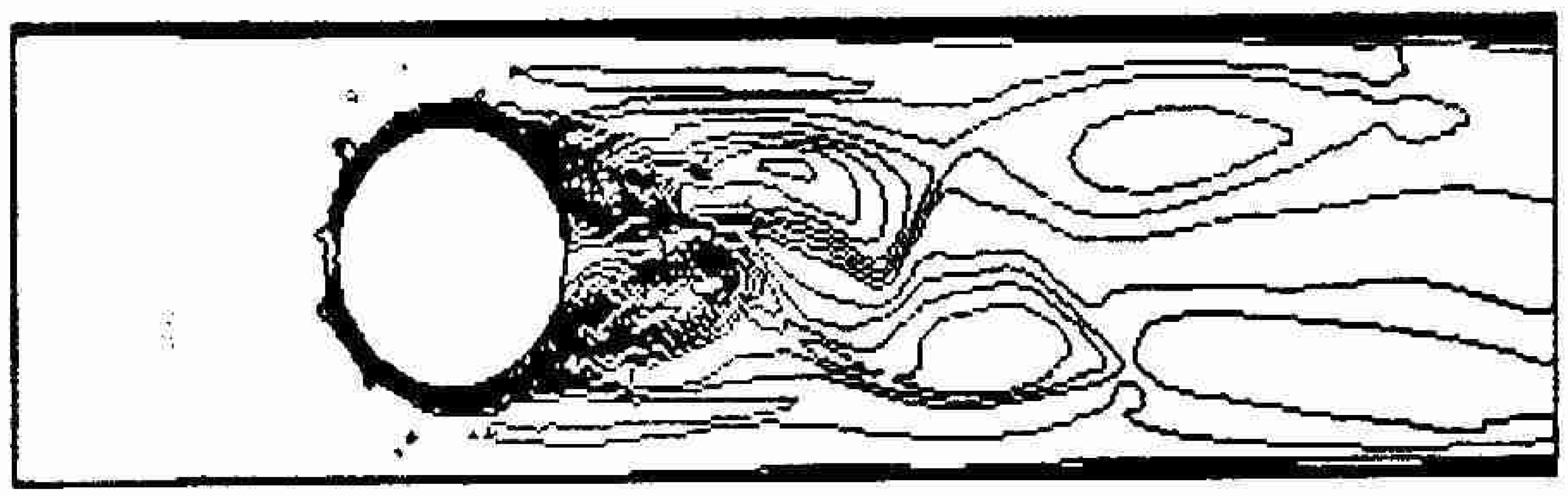
(b) Stream functions for Re=60,000



(b) Vorticity contour for Re=60,000

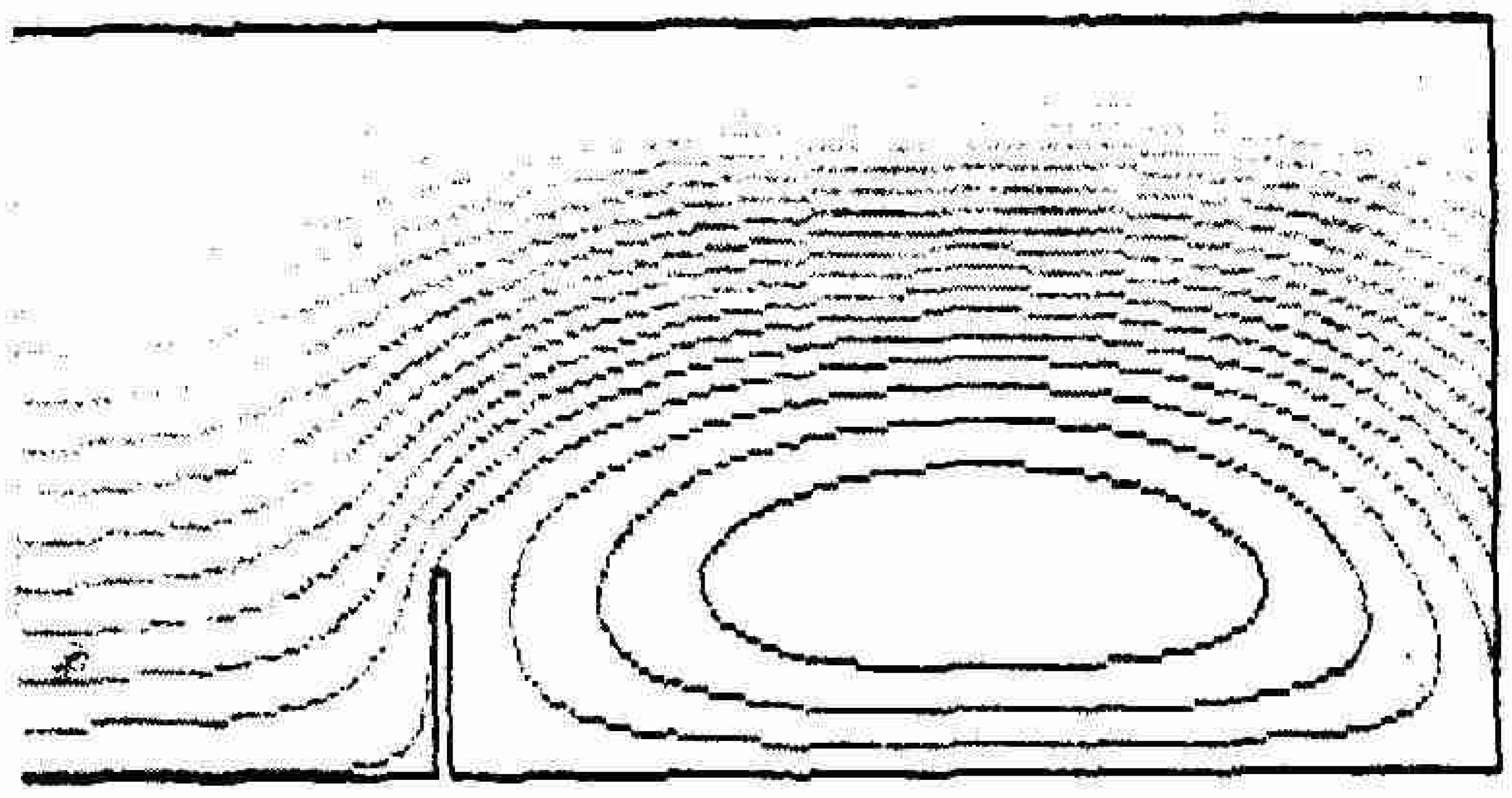


(c) Stream functions for Re=600,000

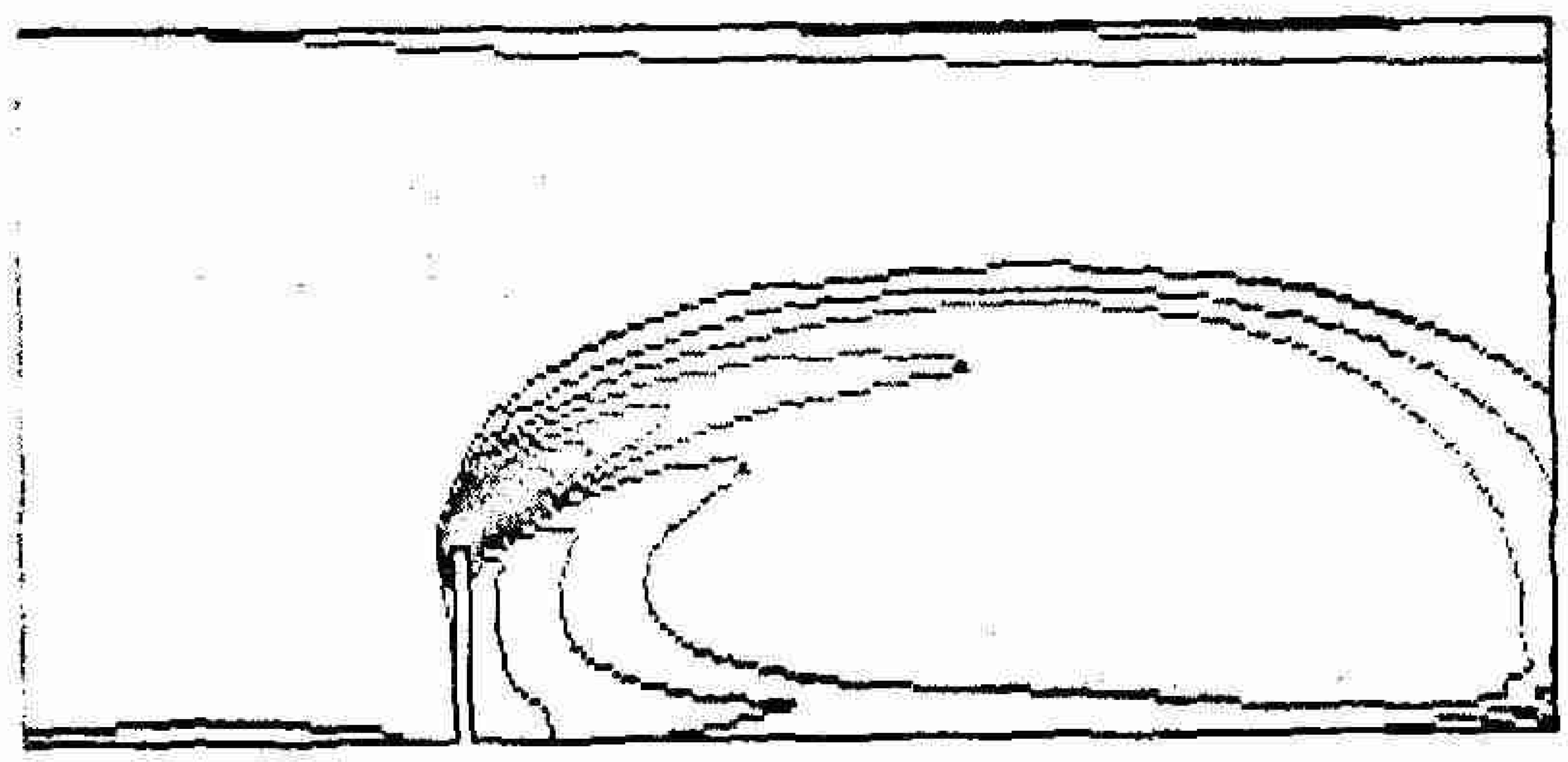


(c) Vorticity contour for Re=600,000

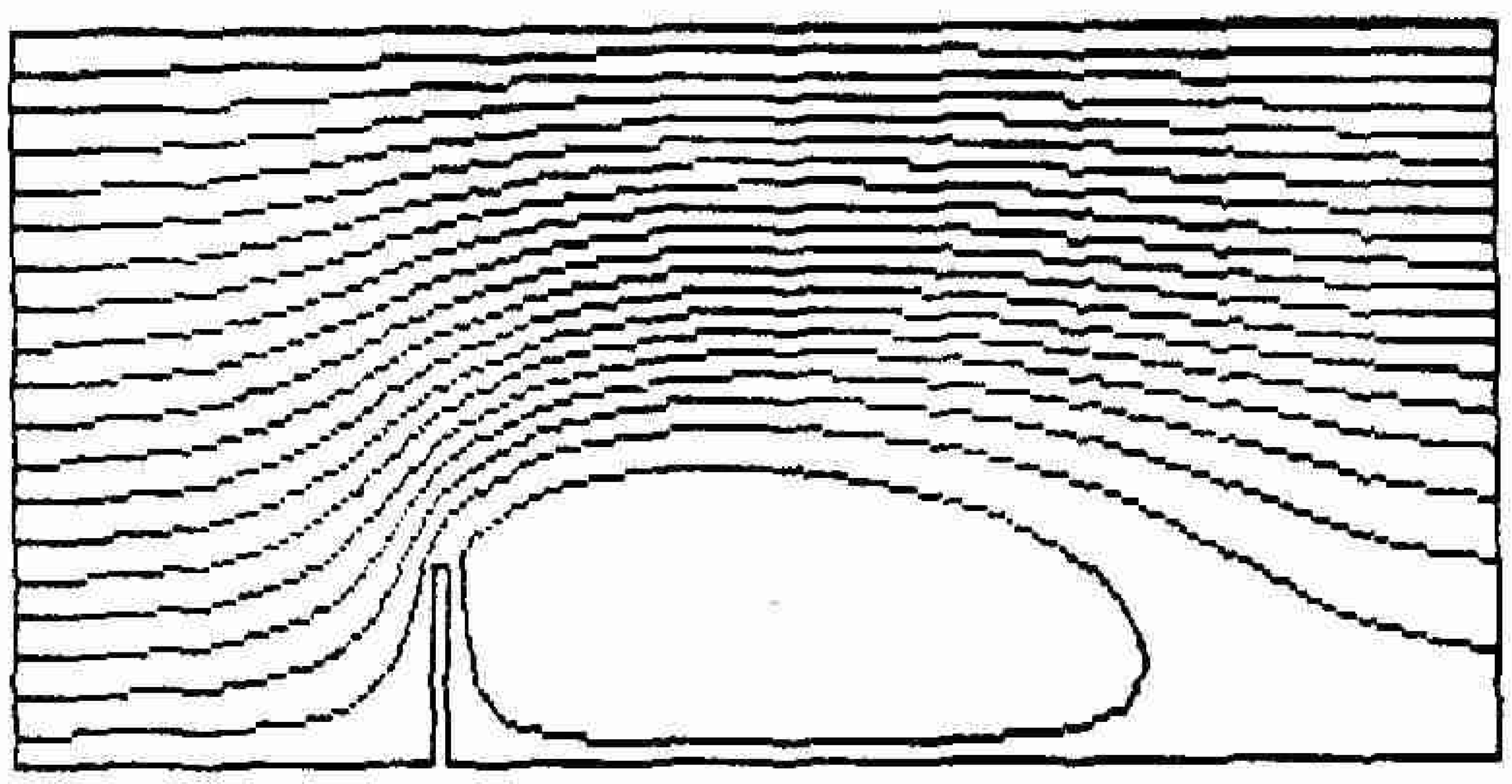
Fig. 5: Stream functions and vorticity contours for flow around a vertical rectangular obstacle for different Reynolds numbers.



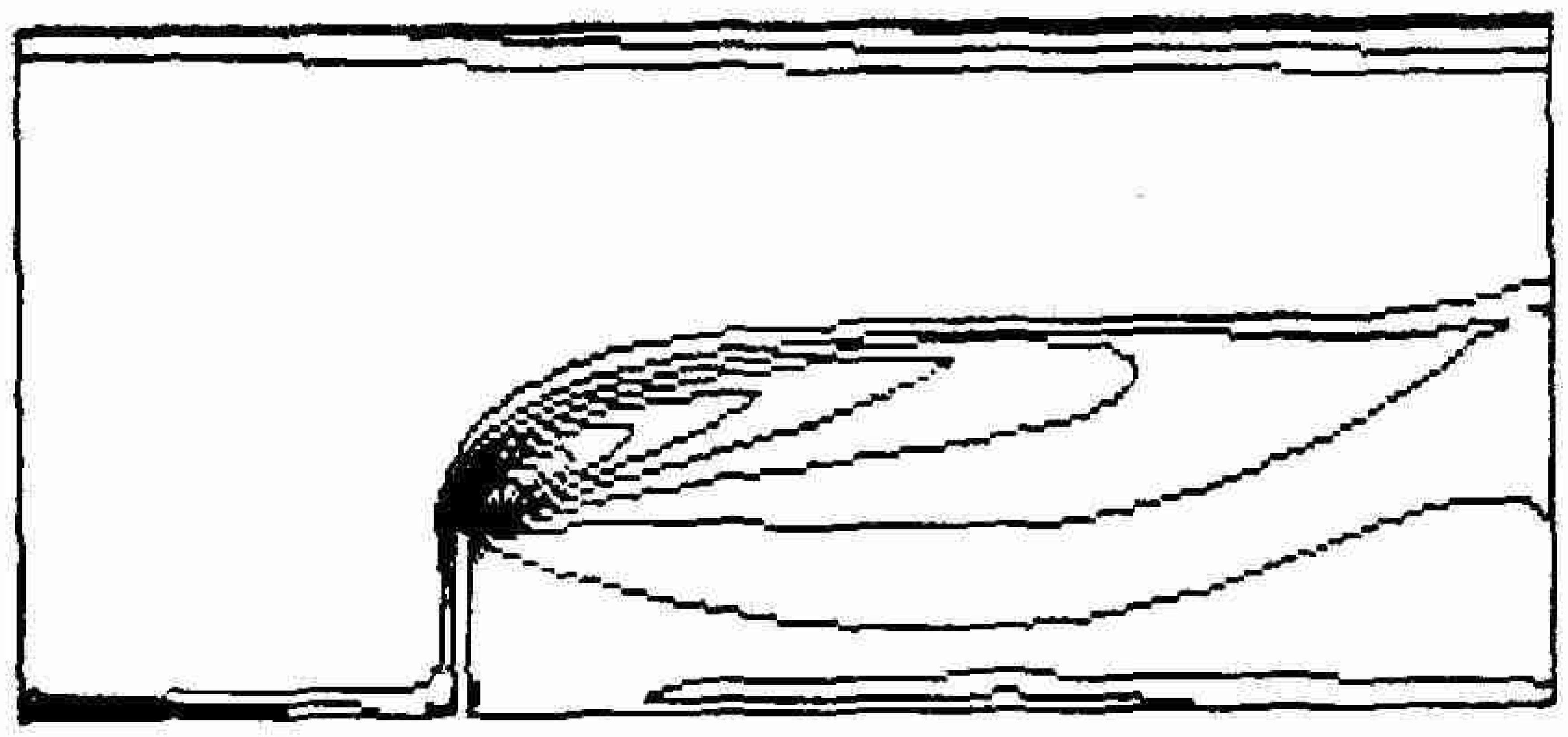
(a) Stream functions for Re=20,000



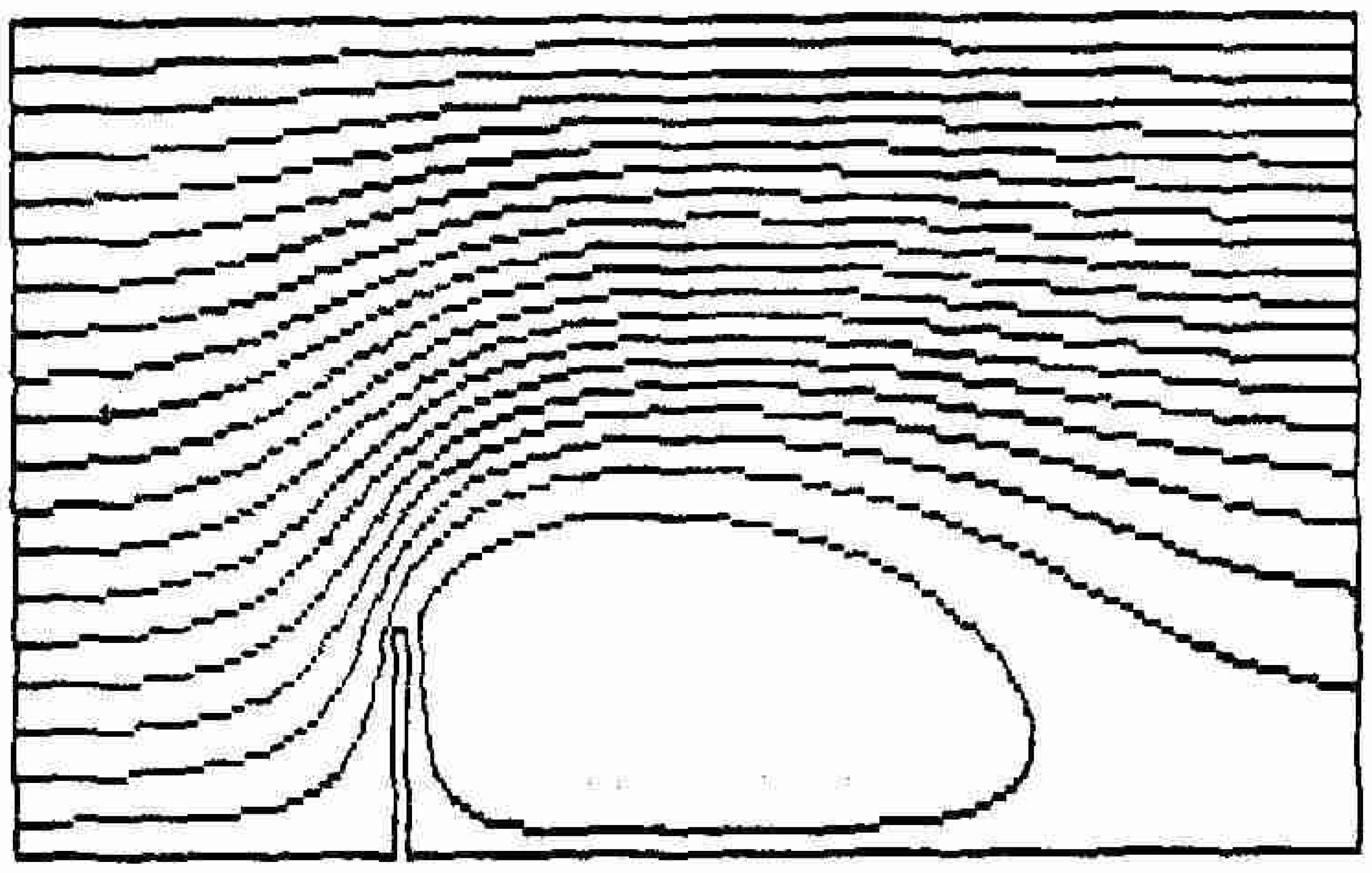
(a) Vorticity contours for Re=20,000



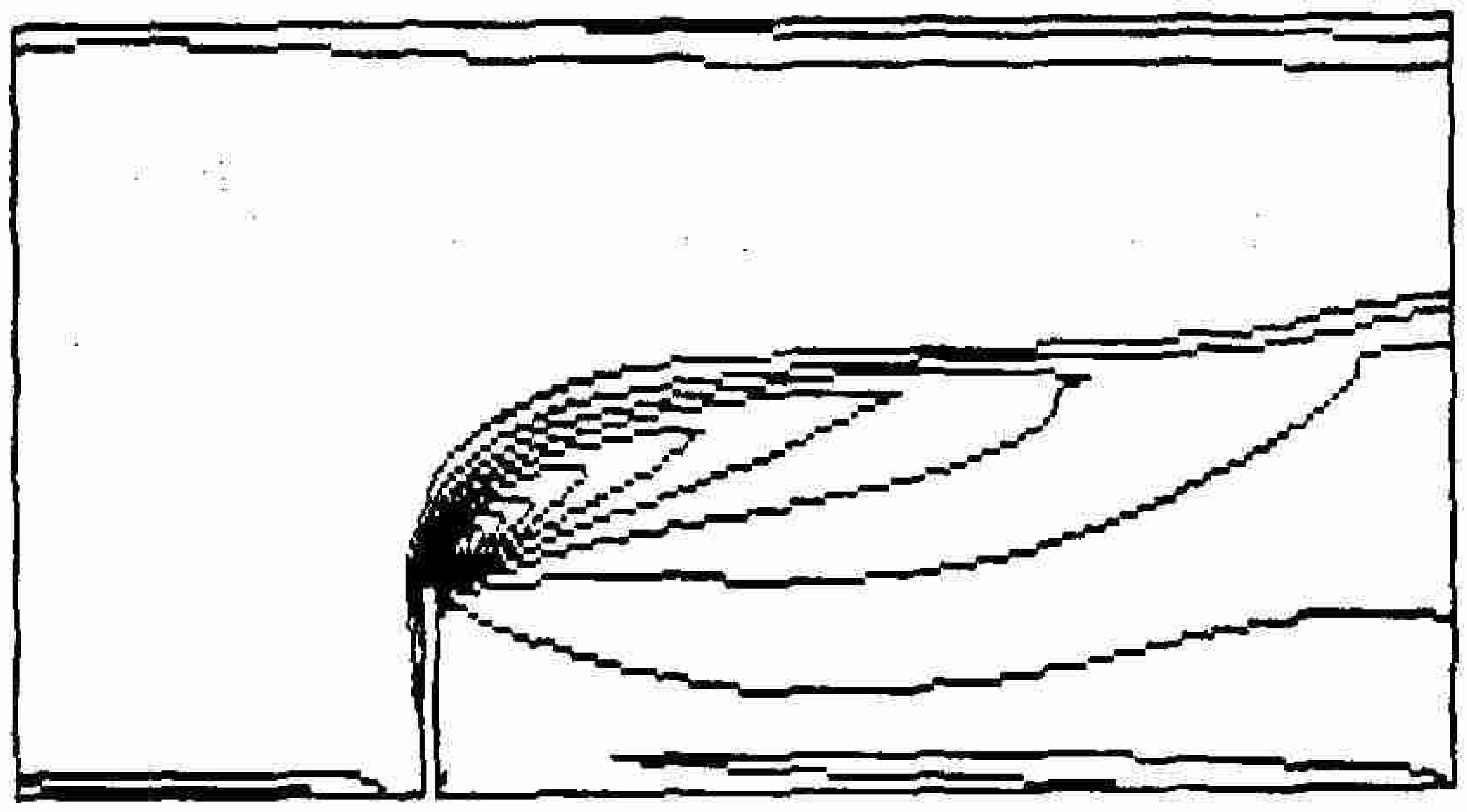
(b) Stream functions for Re=60,000



(b) Vorticity contours for Re=60,000



(c) Stream functions for Re=600,000



(c) Vorticity contours for Re=600,000

Fig. 6: Stream functions and vorticity contours for flow around a vertical rectangular obstacle for different Reynolds numbers.