AIAA-65-3

P 3

MULTIDIMENSIONAL FLUID DYNAMICS CALCULATIONS WITH HIGH SPEED COMPUTERS

by

C. W. HIRT
University of California
Los Alamos, New Mexico

AIAA Paper
No. 65-3

AIAA
2nd Aerospace
Sciences Meeting
NEW YORK, N.Y./JANUARY 25-27, 1965

MULTIDIMENSIONAL FLUID DYNAMICS CALCULATIONS WITH HIGH SPEED COMPUTERS*

C. W. Hirt
Los Alamos Scientific Laboratory
University of California

ABSTRACT

A brief survey is presented of several time dependent numerical methods for multidimensional fluid problems in use at the Los Alamos Scientific Laboratory. Emphasis is placed on the variety of problems which can be treated, as well as on the limitations of the methods used to treat them. No details are given of the specific numerical procedures. Three examples are presented. The first, the interaction of a shock with a bubble, illustrates a type of mixed Eulerian and Lagrangian method of calculation. The second example, a shock moving down a bent channel, illustrates a pure Lagrangian calculation. The third example illustrates an incompressible fluid calculation based on a pure Eulerian method. In the second two examples comparisons are made with experimental data. The presentation of these problems is supplemented with several slides and two short movies.

I. INTRODUCTION

In this paper a brief survey is made of several methods used for the numerical solution of multimaterial and multidimensional fluid dynamics problems. In particular, three methods will be described which are currently in use at the Los Alamos Scientific Laboratory. One additional method will be described by T. D. Butler in another paper.

The aim of this paper is twofold. First, it is an introduction for persons with no previous experience in time dependent numerical methods, to the variety of problems which have been successfully treated. Second, it is hoped that this paper will acquaint those persons with some experience in time dependent numerical methods with a wider class of methods, together with their advantages and disadvantages.

To satisfy the introductory nature of this paper we discuss three quite different problems. The first problem deals with a type of two material calculation in which a shock encounters a bubble in the interior of an explosive material. The second problem involves the calculation of a shock passing down a bent rectangular channel, and the third problem considers the wake of a flat plate impulsively accelerated in a viscous incompressible fluid.

In each of these problems a different numerical approach has been used. In the first problem the explosive material is represented by a system of particles which move relative to a fixed (Eulerian mesh). In the second problem the fluid is again represented by a system of particles, but in this case there is no mesh, the particles are moved relative to one another according to appropriately chosen pairwise forces. This is a pure Lagrangian method. In the third problem, which involves an incompressible fluid, fluid quantities are calculated at the boundaries of a fixed Eulerian mesh. No use is made of particles in this method so this is a pure Eulerian method.

II. DISCUSSION OF THE PROBLEMS

A. Hot Spot Formation

Motivation for this problem arose in connection with experimental studies of shock waves in explosive materials. These studies indicated that a shock, which would not detonate a homogeneous explosive, could detonate the same explosive if it was sufficiently inhomogeneous. The following model situation was devised to study this phenomenon.[1] A plane shock moves along the axis of a cylinder of nitromethane, the shocked nitromethane having a temperature of $1200^{\circ}K$ which is insufficient to produce an immediate detonation. The shock then encounters a spherical void in the interior of the explosive. The interaction of the shock with this "inhomogeneity" produces local hot spots with temperatures exceeding $1400^{\circ}K$. Since nitromethane is highly exothermic above $1400^{\circ}K$ these hot spots can initiate a detonation.

The theoretical studies based on this model were performed using the Particle-In-Cell method,[2] i.e., particles moving relative to an Eulerian mesh. This method is particularly suited to problems involving material interfaces. The disadvantage of using particles is that density and pressure profiles show fluctuations that are often undesirable.

The calculations of hot spot formation with the Particle-In-Cell method were

*This work was performed under the auspices of the U. S. Atomic Energy Commission.

performed on an IBM 7030 computer. A movie has been prepared of the calculated flow configurations. Each frame of the movie was made directly from the output of the computer through an SC 4020 microfilm recorder. The first half of the film shows the collapse of the bubble when chemical reactions are not allowed. Regions of the flow with temperatures exceeding 1400°K are indicated separately. The second half of the film shows a similar calculation in which an Arrhenius type chemical reaction is allowed. In this case a detonation is seen to initiate at the hot spots.

B. Shock Passage through a Bent Channel

The second problem we consider arose in connection with a shock tube study made by Dr. H. Reichenbach at the Ernst-Mach-Institute. Dr. Reichenbach studied a shock passing down a rectangular channel resembling the letter z, having two right angle bends, Fig. 1. Numerical calculations simulating his experiments were undertaken at Los Alamos.[3] Comparisons between the numerical calculations and Dr. Reichenbach's experiments were made through the pressure histories at two points in the channel (see Fig. 1).

To handle this problem a pure Lagrangian representation was used for the fluid. In this method fluid elements are represented by particles interacting with neighboring particles through appropriately chosen pairwise forces (the Particle-And-Force method). Since this method does not use a mesh it is particularly suited to problems with complicated boundaries or problems involving large fluid distortions. On the other hand, pressures are determined from the forces of neighboring particles. In order to reduce fluctuations it is necessary to have a large number of particles. The particle number, however, is limited by computer memory size and the computer time available.

The comparison between Dr. Reichenbach's pressure measurements and the calculated pressures at two points in the channel is shown in Fig. 2. The dashed lines represent an average of the experimental pressure histories obtained from pressure transducers. It will be noted that the agreement with respect to shock arrival time and subsequent pressure history is excellent.

Some comments are in order concerning other numerical methods which could be used for this problem. The previously discussed Particle-In-Cell method has difficulty with problems in which stagnation regions occur. This difficulty is associated with the

natural fluctuations present in a particle system. Small fluctuations in a stagnation region propagate as spurious signals. This difficulty can be reduced, however, by using a pure Eulerian system which calculates a continuous mass flux across mesh boundaries, instead of the number of discrete particles which cross.[4] The pure Eulerian method reduces density fluctuations and consequently it is less affected by the presence of stagnation regions. A calculation simulating Dr. Reichenbach's experiment has been performed using the pure Eulerian method without particles. The results agree very nicely with the Lagrangian calculation. It should be noted, however, that the pure Eulerian method is not suited for problems involving more than one material. This arises from the difficulty of defining material interfaces.

C. Vortex Street Development

While the previous two examples concerned compressible flow situations the third example illustrates the numerical solution of an incompressible viscous fluid problem.[5] This problem is a study of the time evolution of the wake of a flat plate impulsively accelerated along a channel. The channel height was six times the plate height. Numerical calculations were performed for Reynolds numbers between 15 and 6000.

The calculational method is based on a pure Eulerian system, but differs considerably from the methods previously described. Solutions are obtained at points on a fixed Eulerian mesh for two flow variables, the stream function and the vorticity function. The method of solution consists of solving the appropriate finite difference form of Helmholtz's equation for the vorticity. This solution is then used for the source term in a Poisson equation which governs the stream function. A solution of Poisson's equation is obtained by a convergent iteration method. The cycle is then repeated using the new stream function to calculate velocities which appear in the time advanced vorticity equation. Complete details of the method of solution have been discussed in the literature.[6]

A series of calculated streakline patterns are shown on the right hand side of Fig. 3, for Reynolds numbers of 0, 25, and 100. The shedding of vortices, which starts at a Reynolds number of approximately 40, is quite apparent in the R = 100 picture.

The left hand side of Fig. 3 shows a system of isothermals at the corresponding Reynolds numbers. In these pictures the rear side of the plate was maintained at a constant temperature (greater than the free stream temperature), and heat conduction terms were included in the basic transport equations. The fluid density is assumed constant in this problem so that the conduction of heat does not affect the fluid dynamics. A recent extension of this method, however, has been developed to treat the classical Bénard problem, the convection of heat between two parallel plates. In the Bénard problem the conduction of heat does affect the fluid dynamics through the gravitational force term.

A comparison between the calculated streaklines behind a flat plate and the experimentally observed streaklines (dye trails) behind a circular cylinder[7] is shown in Fig. 4. Both cases correspond to a Reynolds number of 100.

Additional comparisons have been made for this problem with other available experimental data; for example, with the plate drag coefficient as a function of time and with velocity profiles behind the plate.[5] Furthermore, a movie has been prepared showing the evolution of streaklines and streamlines for a Reynolds number of 200. This movie was assembled in a manner identical to that used for the bubble problem.

REFERENCES

1. C. L. Mader, Los Alamos Scientific Laboratory Report No. LA-3077 (1964).
2. F. H. Harlow, Proc. Symp. Appl. Math., 15, 269 (1963).
3. B. J. Daly, F. H. Harlow, J. E. Welch, and E. Wilson, Los Alamos Scientific Laboratory Report No. LA-3144 (1964).
4. R. A. Gentry and R. E. Martin, LASL Report, in preparation.
5. J. E. Fromm and F. H. Harlow, Phys. of Fluids 6, 975 (1963).
6. J. E. Fromm, Los Alamos Scientific Laboratory Report No. LA-2910 (1963).
7. A. Thom, Proc. Roy. Soc. (London) 141A, 651 (1933).

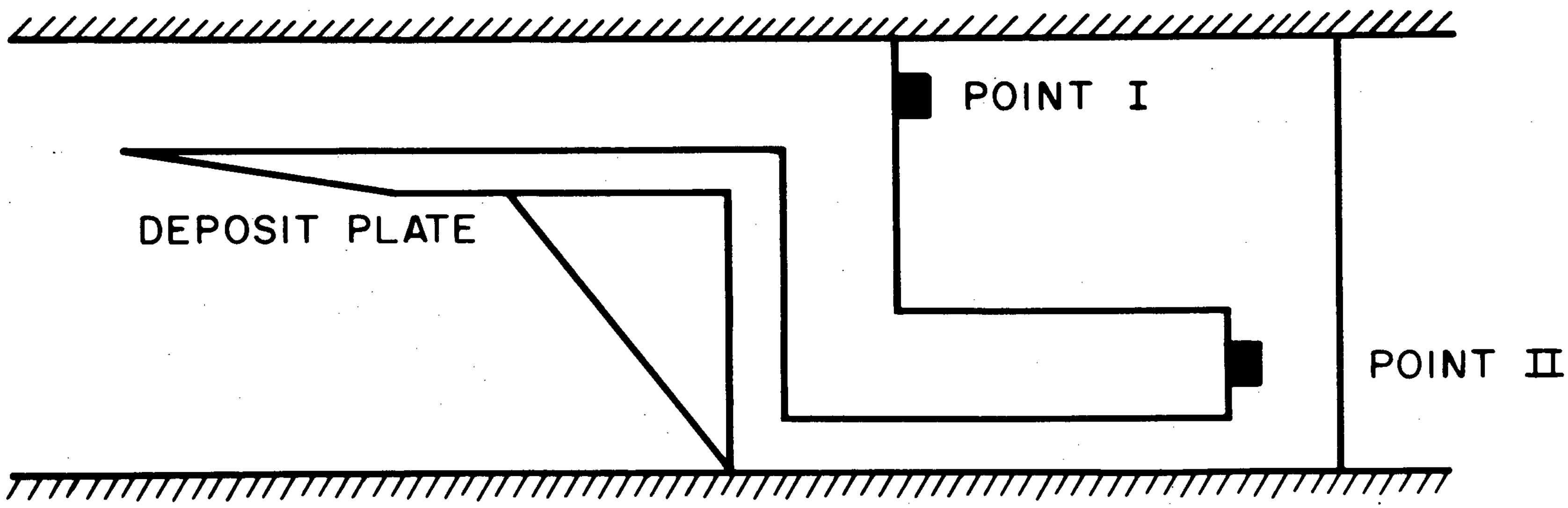

Figure 1. Schematic drawing of rectangular channel. Pressure histories were recorded at points I and II.

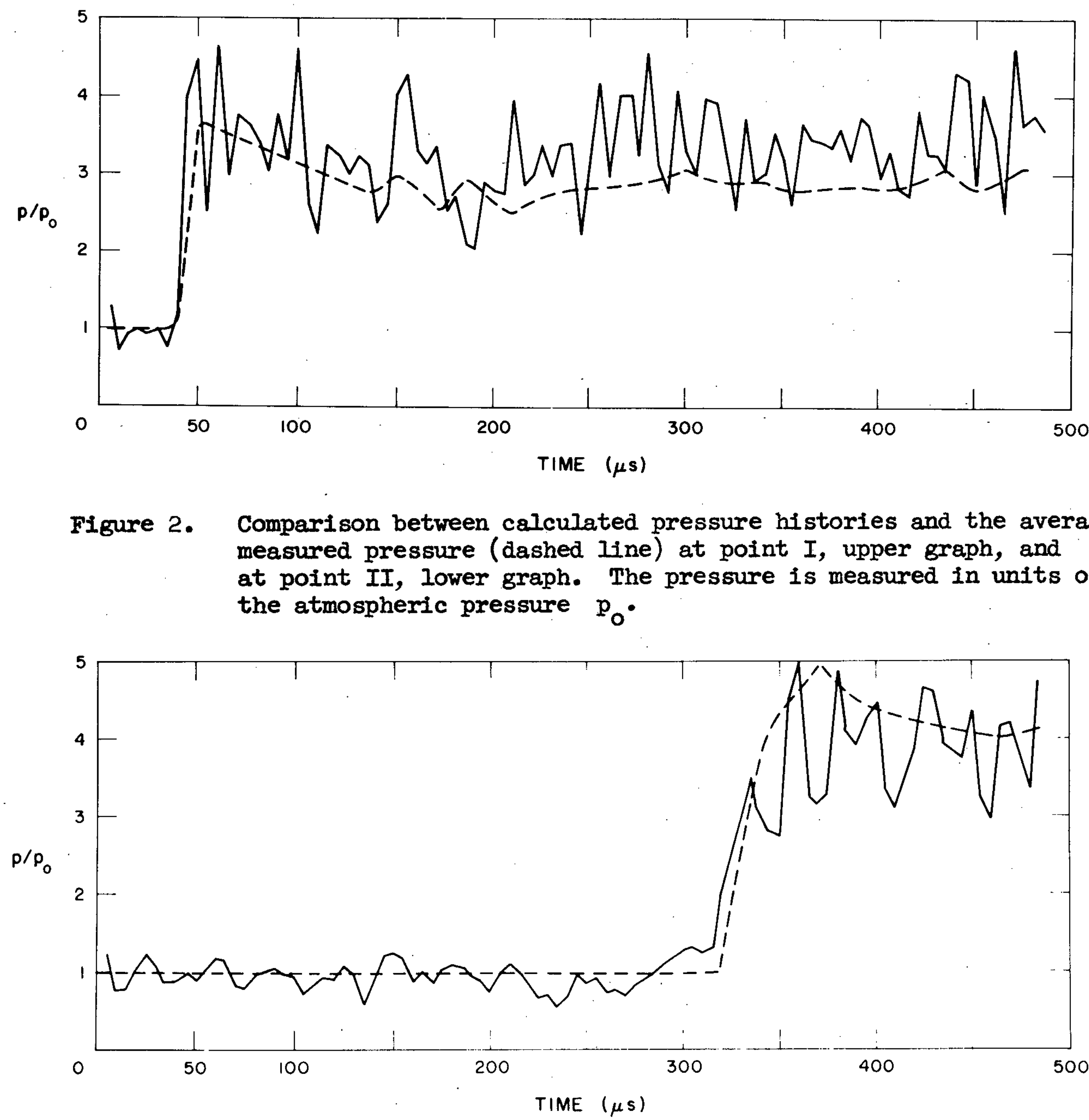

Figure 2. Comparison between calculated pressure histories and the average measured pressure (dashed line) at point I, upper graph, and at point II, lower graph. The pressure is measured in units of the atmospheric pressure p_o.

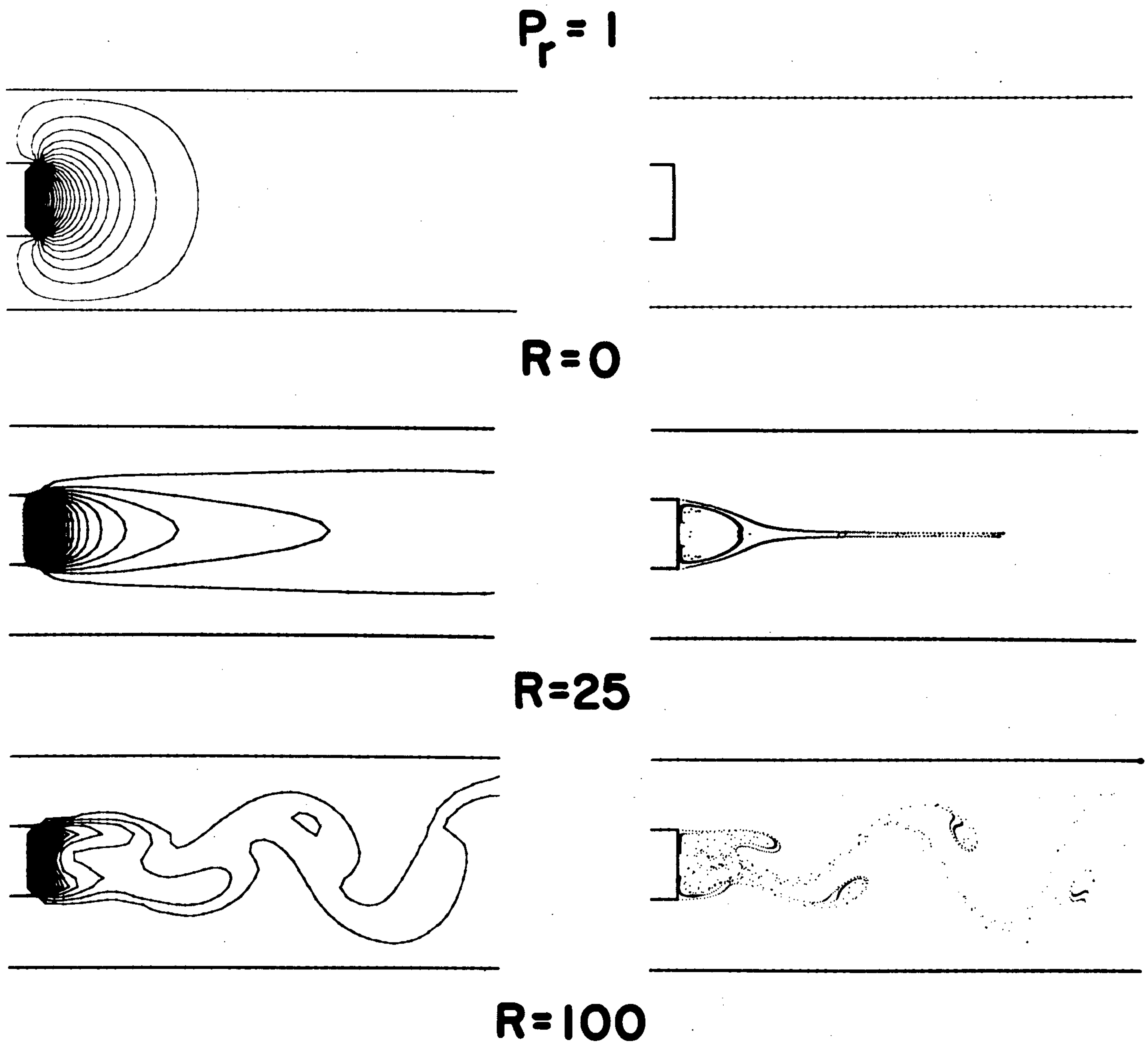

Figure 3. Calculated isothermals on left and streaklines on right for various Reynolds numbers with a Prandtl number of unity.

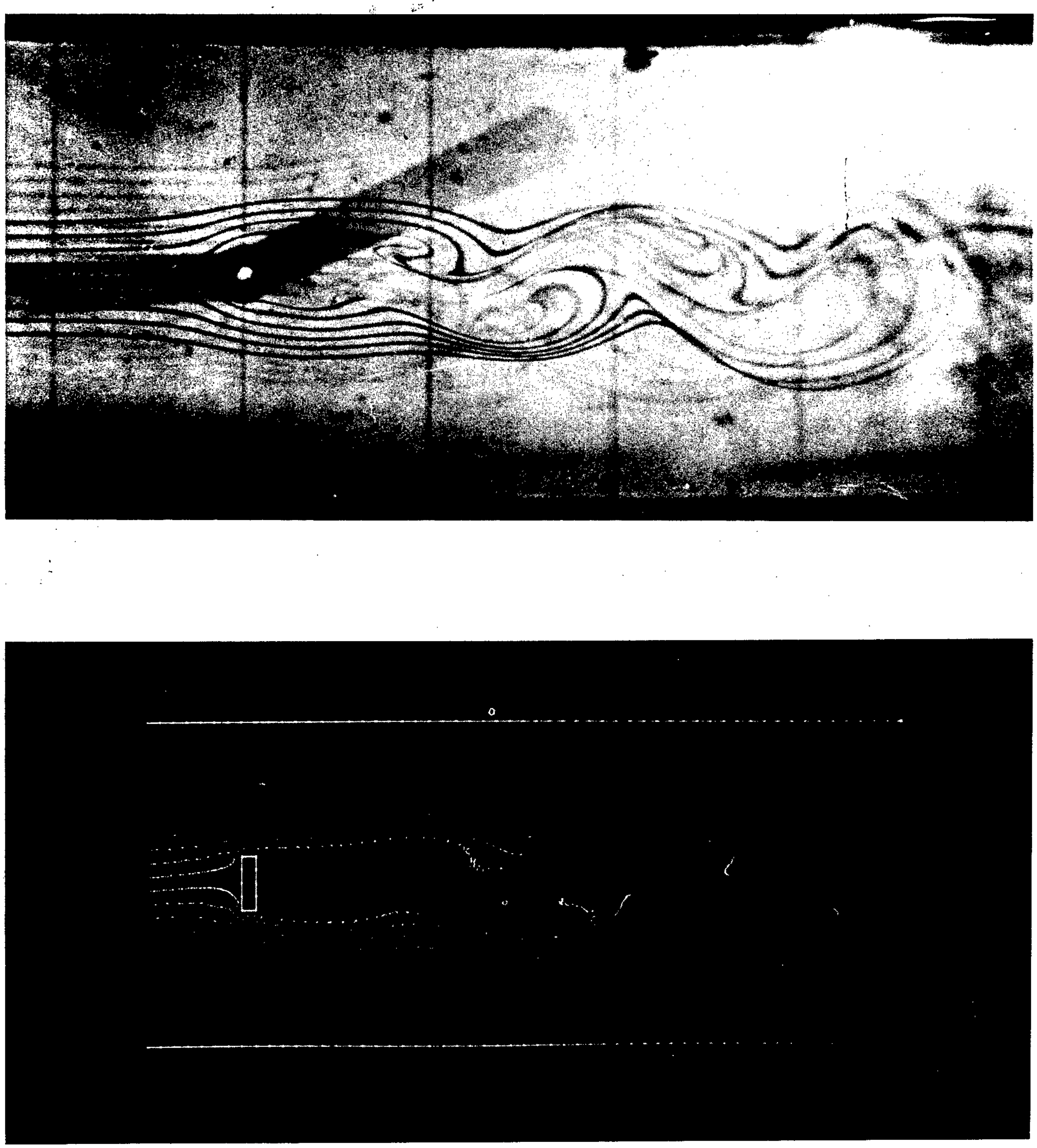

Figure 4. The upper picture illustrates dye trails observed behind a circular cylinder. The lower picture shows the calculated streaklines behind a flat plate. Both pictures correspond to a Reynolds number of 100.