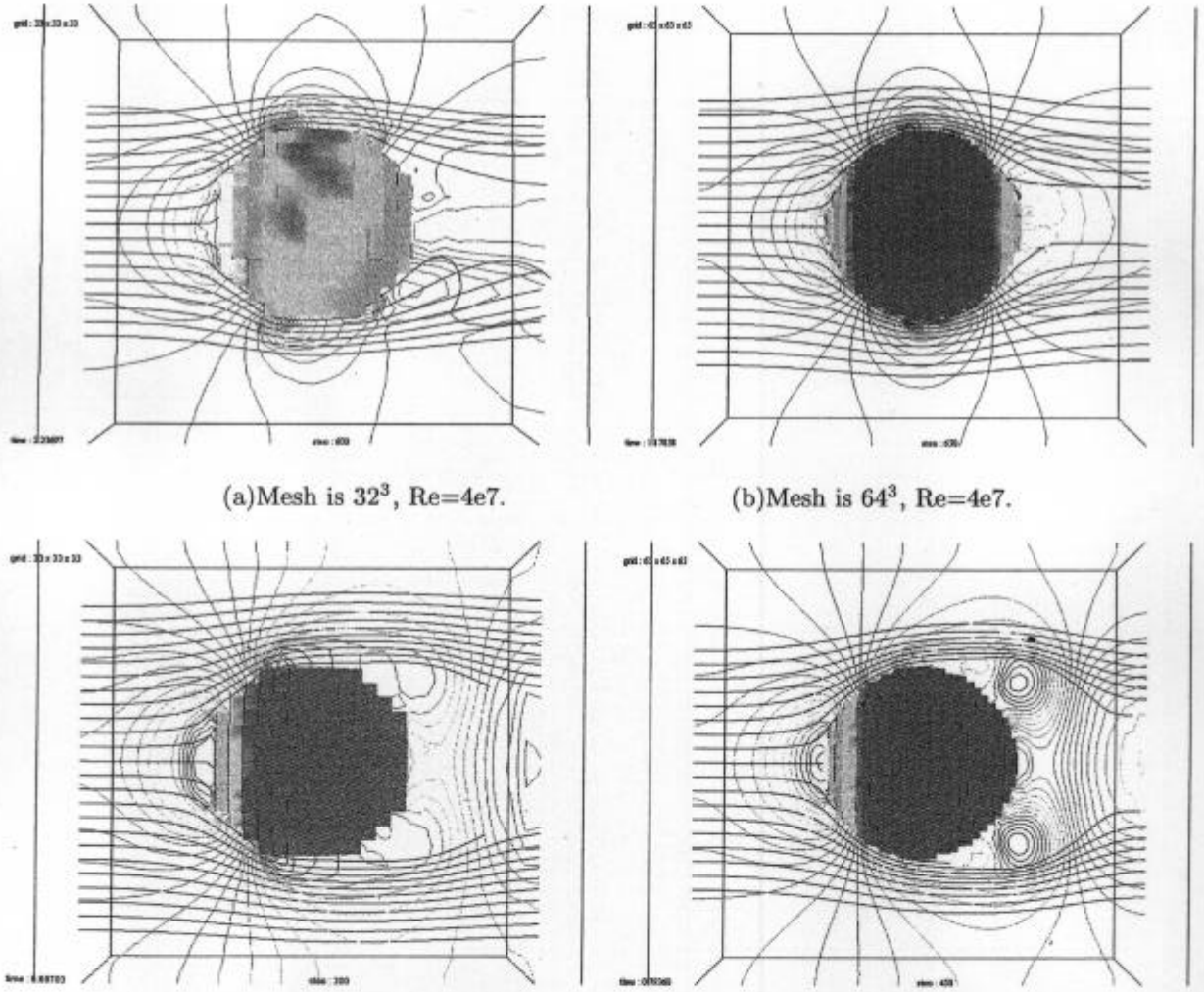
鈍い物体まわりの流れのシミュレーションとそれに働く力

Simulation of a Flow around a Bluff Body and the Force

○ 北村 史郎, 計算流体研, 〒 152-0011 東京都目黒区原町, E-mail : kitamura@icfd.co.jp
小柴 誠子, 日大理工, 〒 101-8308 東京都千代田区神田駿河台, E-mail : satoko@math.cst.nihon-u.ac.jp
桑原 邦郎, 宇宙研, 〒 229-8510 神奈川県相模原市由野台, E-mail : kuwahara@pub.isas.ac.jp
KITAMURA, Siro, Institute of Computational Fluid Dynamics, Haramachi, Meguro-ku, Tokyo 152-0011
KOMURASAKI, Satoko, College of Science and Technology, Nihon Univ., KandaSurugadai, Chiyoda-ku, Tokyo 101-8308
KUWAHARA, Kunio, Institute of Space and Astronautical Science, Yoshinodai, Sagamihara-shi, Kanagawa 229-8510

Impulsively-started flow past a bluff body is simulated by solving the unsteady incompressible Navier-Stokes equations with Cartesian grid system. The multi-directional finite-difference method with third-order upwinding is employed without using any turbulence model. As a result of computations at Reynolds numbers 0.1 – 40,000,000 the separation and the transition to turbulence are investigated in the wake and computed drag coefficients agree well with the experimental values. Also, drag crisis is captured.

鈍頭物体周りの流れとして, 球体周りの流れを計算した. Fig.1(a) ～ (c) にレイノルズ数および格子の違いによる等圧力線図と流線の変化をしめす.



(a)Mesh is 32^3, Re=4e7.

(b)Mesh is 64^3, Re=4e7.

(c)Mesh is 32^3, Re=4e4.

(d)Mesh is 64^3, Re=4e4.

Fig. 1: Pressure contours and stream lines around spheres.

Copyright © 2000 by JSCFD