The Seventh Asian Congress of Fluid Mechanics
Dec 8 - 12, 1997, Chennai (Madras)

NUMERICAL SIMULATION OF FLOW PAST ROTATING & TRANSLATING CIRCULAR CYLINDER

Sanjay Mittal   Manoj T. Nair   Tapan K. Sengupta
Department of Aerospace Engineering, IIT Kanpur, Kanpur 208 016, India.

ABSTRACT   Numerical solution of two-dimensional Navier-Stokes equation by Finite Element (FE) and Finite Difference (FD) method have been obtained and compared with experimental results at Re = 1,000 for flow past an impulsively translated and rotated circular cylinder. The agreement with experimental results is very good. Furthermore the computations performed at Re = 3,800 are reported for the first time giving detailed load history.

1. Introduction

The study of flow past circular cylinder that is rotated and held in a uniform stream has received attention for more than a century starting with the orginal experiments of Magnus [1] examining lift generation by imposed circulation. The rotational to translational speed ratio (α) and Reynolds number based on free stream speed and diameter are the parameters of the problem. Later on this flow has been studied to: i) understand unsteady boundary layer separation, ii) modify the shed vortices in the wake and iii) control the aerodynamic performance by the use of unsteady rotation.

In [1] it is pointed out that at high rotation rate ($\alpha > 2$) there is the possibility of generating the corresponding irrotational velocity distribution as a steady flow of real fluid at large Reynolds number. In this context Tokumaru & Dimotakis [2] in analysing their experimental results observed that the theoretical maximum limit for lift coefficient can be exceeded for high rotation rates. They have provided results for $0.5 < \alpha < 10$ at Re = 3,800. Here we provide some results for this Reynolds number and additional results at Re = 1,000 for which both experimental and computational results exist.

2. Formulation & Numerical Methods

The two-dimensional Navier-Stokes equation representing incompressible flow is solved in primitive variables by FE method [3] and in the stream function and vorticity formulation by FD method [4,5]. The FE method [3] is based on SUPG (Streamline-upwind/Petrov-Galerkin) and PSPG (Pressure-Stabilising / Petrov-Galerkin) stabilizing technique using equal-order-interpolation velocity-pressure element. The nonlinear equation systems resulting from the discretization are solved using Generalised Minimal RESidual (GMRES) technique with diagonal preconditioners. The FD method [4,5] is based on the iterative solution of stream function equation by Alternating Direction Implicit (ADI) & Modified Strongly Implicit Proceedure

(MSIP) and the vorticity transport equation by higher order upwinding schemes [4,5] which stabilize the computation implicitly for high Reynolds number flows.

3. Results & Discussion

The FE method uses 12,000 elements approximately and the FD method uses (250X400) points. For the explicit FD method the time step chosen is t = 10^{-4} and for the FE method this is about 10^{-2}. The FD methods uses a O-grid topology with the outer boundary placed at 26 diameters and the first grid line is 10^{-5} diameter away from the surface-while in FE method a rectangular outer boundary is used with the downstream boundary located 25 diameters away. The pressure in the FD method is obtained by the solution of Poisson equation by Conjugate Gradient method[5]. In Fig. 1 we have compared the computed solutions for Re = 1000 & $\alpha = 3$ with the experimental results of [6] at t = 2 & 3. The level of agreement between the two sets of computations and with the experiment is excellent. In Fig. 2 we have shown the load histories of the present FE computations for Re = 1000 and α = 1, 2, 3 & 5. These results are different than what has been shown in [7] in the following respect: i) The mean and fluctuating components of c_l & c_d respectively are higher and lower in the present computations, ii) The sharp change in drag value as α crosses 2 as noticed in [7] (Fig. 26) is not observed here and iii) The higher α cases at 3.0 and 5.0 for the present computations did not reveal very high frequency fluctuations as noticed in [7]. This could be due to the order of the present method, as well as the effect of round-off error in hybrid vortex method that is used in [7].

The FE method is furthermore checked with the FD method in Fig. 3 for Re = 3,800 & α= 2. where computed streamlines at t = 6.5 are shown. For this high Reynolds number the flow topology is far too complicated compared to the earlier case. The excellent comparison establishes consistency of the two methods. In Fig. 4, we have shown the load histories along with the FD results for α = 0.5, 1 & 2 cases. Note that the time scales differ by a factor of two in these formulations and the final real time upto which computations are done is identical. The match between the FE & FD methods for the amplitude and phase for c_l & c_d for $\alpha = 2$ is excellent which gives confidence in these results.

References

[1] Batchelor G, An Introduction to Fluid Dynamics, CUP, (1988).
[2] Tokumaru PT & Dimotakis PE, JFM, 255, 1-10, (1993).
[3] Mittal, S & Tezduyar, TE, IJNMF, 15, 1073-1118, (1992).
[4] Sengupta, TK & Sengupta, R, J. Comp. Mech., 14, 4, 298-310, (1994).
[5] Nair MT, Sengupta TK & Chauhan US, Accepted for publication in Computers & Fluids, (1997).
[6] Badr, HM, Coutanceau, M, Dennis, SCR & Menard, C, JFM, 220, 459-484, (1990).
[7] Chew, YT, Cheng, M & Luo, SC, JFM, 299, 35-71, (1995).

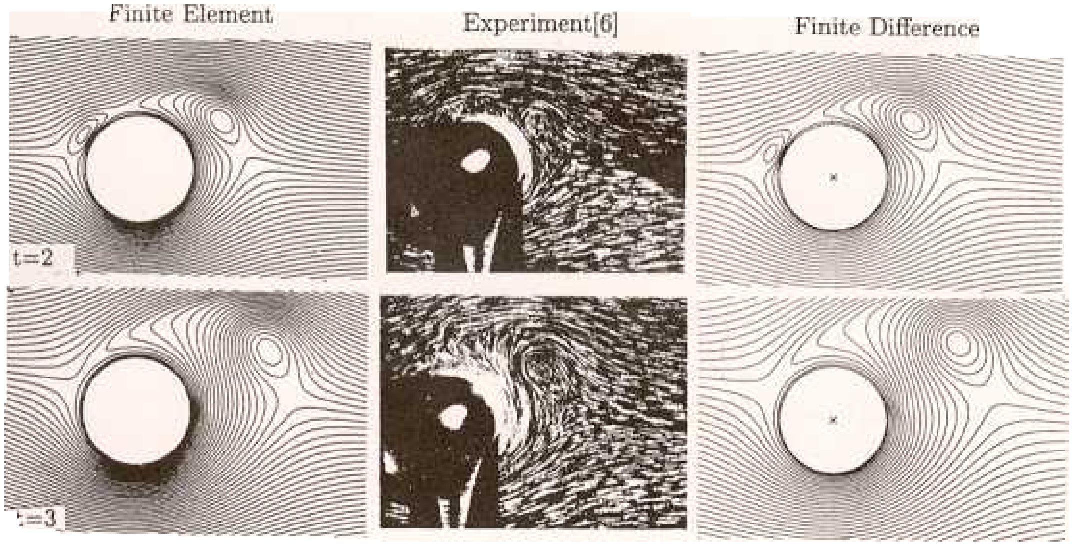

Fig. 1. Computed & Experimental streamlines for $Re = 1000$ $\alpha = 3$.

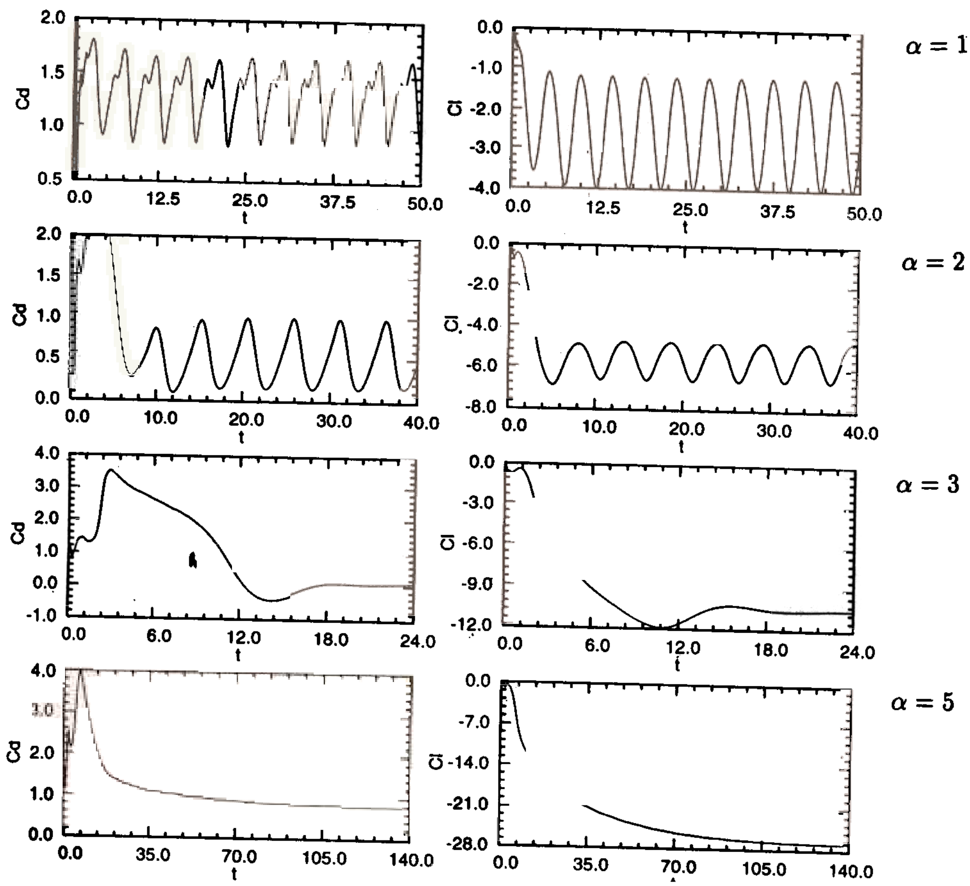

Fig. 2. Load history for $Re = 1000$.

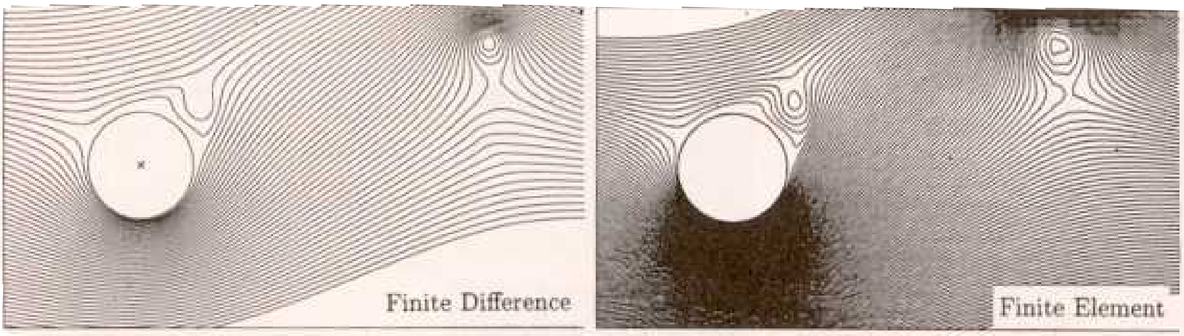

Fig. 3. Computed streamlines for $Re = 3800$, $\alpha = 2$ at t=6.5

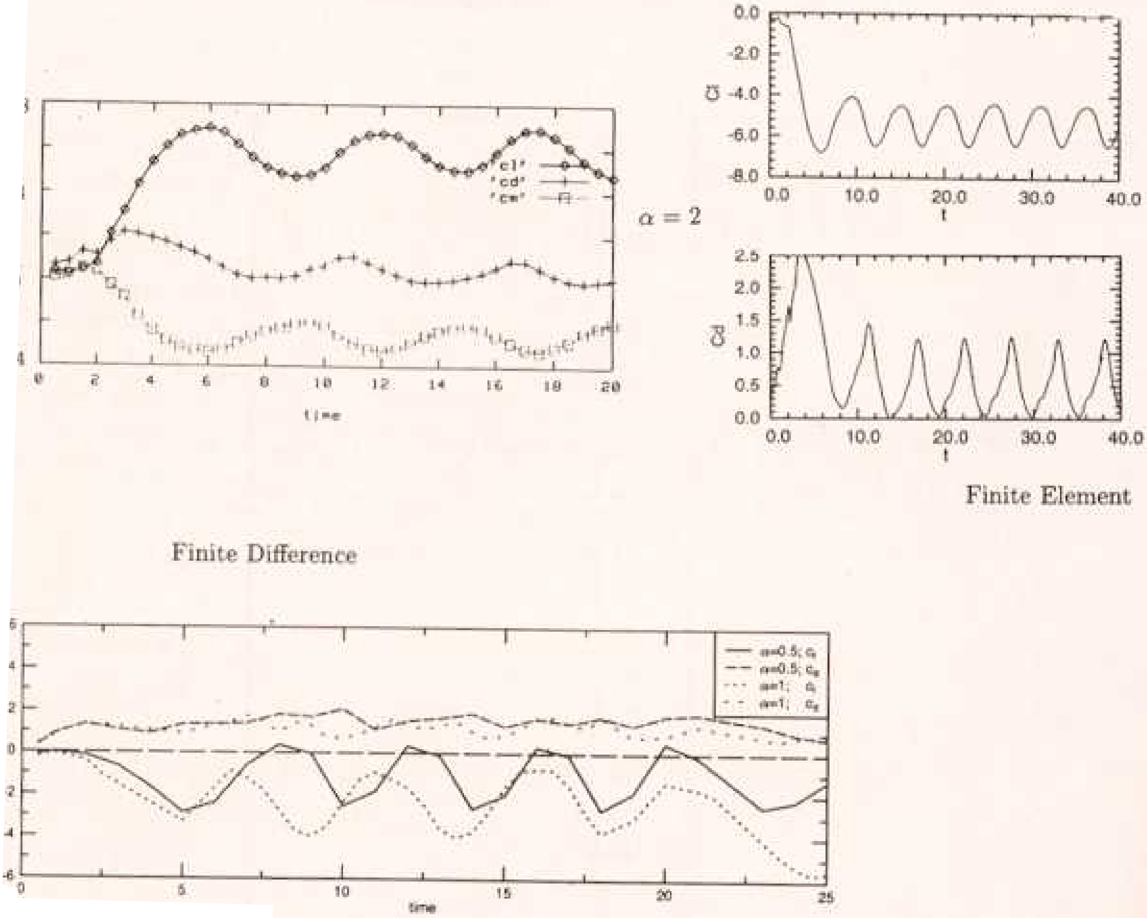

Fig. 4. Load history for $Re = 3800$ by FD & FEM.