Int. Conference on Boundary and Interior Layers
BAIL 2006
G.Lube, G.Rapin (Eds)

Numerical Simulation of High Sub-critical Reynolds Number Flow Past a Circular Cylinder

Wan Saiful Islam* and Vijay R. Raghavan**

Faculty of Mechanical Engineering
Kolej Universiti Teknologi Tun Hussein Onn
86400 Parit Raja, Malaysia
*wsaiful@kuittho.edu.my
**vijay@kuittho.edu.my

1. Introduction

Few areas in fluid mechanics have received more attention than that of flow past a bluff body. In particular, flow across a circular cylinder in unconfined and confined flow is a classical problem, and has been studied experimentally, visually and numerically. Studies on flow characteristics of tubes in cross flow have long been of interest due to their wide use in engineering applications. Though the geometry is very simple, the flow pattern in the vicinity of the cylinder is full of variety. This problem has not yielded to closed form analytical solution except at very low Reynolds numbers. The large volume of literature using a variety of numerical approaches viz., finite difference, finite element and finite volume methods associated with this particular subject attests to the importance as well as the complexity of the fluid flow involving adverse pressure gradients, separation, eddy shedding, recirculation and reattachment. However, there is room for improvement in the agreement with experiments that has been obtained hitherto. A numerical solution that gives good agreement is also likely to be useful for benchmarking existing codes and new ones that may be written.

2. Present Study

In the present study simulations are performed for unsteady, two-dimensional (2-D) flow past a circular cylinder in a confined duct with appropriate blockage ratios. At Reynolds numbers as high as 100,000 the numerical solutions obtained agree remarkably well with experiments, not only in the global sense in the form of C_D, but also locally in terms of pressure distribution. These results might serve as a benchmark for validating CFD codes.

3. Earlier Attempts at Modeling the Flow

The number of reported numerical investigations was at a low pace in the first half of the 20th century. However, the rate has increased tremendously in the last few decades with the improvement of numerical algorithms accompanied by powerful digital computers. In earlier attempts to establish benchmark solutions and to obtain agreement with published data over a range of Reynolds numbers, various turbulence models including large eddy simulation (LES) had been considered and both 2-D and 3-D had been tried. The results were obtained in the form of general appearance of the wake flow, examination of the velocity magnitudes in the near-field and far-field, eddy frequencies, Strouhal numbers and detailed local distributions of pressure. However, most of these authors have carried out their work in the more amenable Reynolds number range of 40 to 1000. As one goes to higher Reynolds numbers, results are seen to deviate more and more from experimental results.

4. Background Literature

The reported numerical investigations covered many aspects including dimensionality of the model (2-D or 3-D), formulation of the flow variables, time dependence (steady or unsteady), compressibility of the flow, turbulence modeling, and discretization schemes. Beginning with the early works of Thom (1933), Takami & Keller (1969), Thoman & Szewczyk (1969) and Son & Hanratty (1969) have attempted to present results to as high as Re = 10^5, but succeeded only up to the order of Re = 200 in terms of agreement with experiments. Since then many researchers have proposed a variety of numerical schemes to improve the accuracy of the results.

With an objective to overcome the difficulty in formulating for incompressible flow, Kawahara and Hirano (1983) have considered a slight compressibility and simulated the flow up to Re = 1.5×10^5. However, the authors observed a considerable discrepancy particularly in the computed C_p results. Braza et al. (1986) simulated the flows for Re up to 1000 by introducing numerical perturbations to trigger the vortex shedding. Song and Yuan (1990) modeled vortex shedding by imposing a hybrid boundary condition over the periphery of circular cylinder. Without introducing any artificial disturbances to trigger the vortex shedding, Franke et al. (1990) simulated the flows up to Re $\leq$ 5000 using third order QUICK scheme for spatial discretization and fully implicit first order Euler scheme for temporal discretization. However, for flows of Re > 1000 the results deviate from the measurements, and the authors attributed this to the beginning of the influence of random fluctuations.

Interestingly, a few reports {Tamura et al. (1990), Kakuda and Tosaka (1993) and Kondo (1993)} have appeared on the simulation of flow past a circular cylinder at high Reynolds numbers ($10^4 < Re < 10^6$) without using any turbulence model. From these works it is noticed that the 2-D simulations have failed to produce accurate results. In the 3-D simulations, a periodic boundary condition was imposed in the span wise direction.

Murakami (1993) and Rodi (1993) have used LES to simulate the flow past bluff bodies and compared the accuracy of LES against the well-known turbulence models viz., k-ε eddy viscosity model, algebraic stress model, etc. They summarized their work by saying that the LES provides greater estimation accuracy than the other turbulence models but requires much more CPU time and memory. With an objective to investigate the numerical and modeling aspects such as three-dimensional effects, discretization schemes, subgrid scale modeling and grid refinement, Breuer (1998) has solved the flow past a circular cylinder at Re = 3900 using LES and finite volume method. Again he also considered periodic boundary condition in the span wise direction.

From the literature survey it is observed that a majority of the numerical studies of this problem have been reported in the last 15 years. This is due to the improvement in the numerical schemes and availability of powerful digital computers. It is seen that satisfactory numerical results in the high subcritical Reynolds numbers are scant. Many have not reported the local distribution of pressure on the periphery of cylinder. The literature shows that there is scope for improvement in the agreement with experiments that has been obtained hitherto.

5. Numerical Model

This paper presents results of numerical calculation of the unsteady Navier-Stokes equations for laminar two-dimensional flow past a circular cylinder using a general-purpose CFD code, FLUENT. Only a brief overview of the particular approach employed for this simulation is discussed here.

The schematic diagram of the flow past a circular cylinder is shown in Fig.1. Boundary conditions are specified on its respective edges. Uniform velocity boundary condition is assumed at the upstream boundary, the no-slip condition is imposed on the cylinder as well as on the sidewalls. At the outlet, a zero diffusion flux condition is employed for all variables followed by an overall mass balance correction. To minimize the effect of the outflow boundary condition on the flow in the vicinity of the cylinder, the computational domain is extended to 16 cylinder diameters downstream of the center of the cylinder. The computational domain is discretized with a hybrid mesh, which comprises of triangular and quadrilateral cells, as shown in Figures 2 and 3. Quadrilateral cells are created around the cylinder to study the velocity gradients of the boundary layer more accurately, with a large number of grid points in the boundary layer.

6. Solution Approach

$$\frac{\partial u_i}{\partial x_i} = 0 \; ; \qquad \text{and} \qquad \rho\left(\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j}\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}$$

where $\tau_{ij} = \left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] - \frac{2}{3}\mu \frac{\partial u_j}{\partial x_i}.\delta_{ij}$ is the stress tensor.

Using the control-volume-based technique the governing equations are integrated about each control volume to obtain algebraic equations, which are then solved numerically. The time dependent equations for conservation of mass and momentum are discretized in both space and time. By using second order upwind scheme the convection terms are interpolated from the cell center values. The diffusion terms are central-differenced with second-order accuracy. The discretized transport equations are linearized and subsequently solved using a point implicit (Gauss-Seidel) linear equation solver in conjunction with an algebraic multigrid (AMG) method. The use of this multigrid scheme greatly reduces the number of iterations and the CPU time required to obtain a converged solution, particularly when the model contains a large number of control volumes.

The momentum and continuity equations are solved sequentially and in this sequential procedure, the continuity equation is used as an equation for pressure. The Pressure-Implicit with Splitting of Operators (PISO) pressure-velocity coupling scheme, part of the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) family of algorithms, is used for introducing pressure into the continuity equation. PISO is based on the higher degree of the approximate relation between the corrections for pressure and velocity. The temporal terms are discretized using backward differences of first order accuracy and subsequently the functional value is evaluated at the future time level using a fully implicit time integration scheme. The advantage of the fully implicit scheme is that it is unconditionally stable with respect to time step size.

During a simulation, solution convergence is monitored dynamically by checking residuals, forces, and surface integrals against the elapsed time. Monitoring the forces is often useful when the forces converge before the residuals converge, and thus time can be saved by terminating the simulation earlier than the residual meets the convergence criterion.

7. Comparison of Present Results

Simulations are performed for unsteady, two-dimensional, incompressible flow in the Reynolds number range of practical importance. Properties of air (at 300K) are used to determine the Reynolds numbers and drag coefficients. The comparisons of numerical solutions have been carried out for the stated combination of conditions of each experiment. Fig.4 shows the time averaged pressure distribution of flow past a circular cylinder at various combinations of conditions viz., Reynolds number and blockage ratio. These numerical results are in very good agreement with the available experimental data.

The waviness in pressure distribution on the leeward side of the cylinder, in the approximate range $\phi = 130$ - 150^0, also noticed in experiments, is explained by the oscillation of secondary eddies on the periphery of the cylinder wall. Minor differences between the simulated C_p values and experiments can be attributed to the unknown process of time averaging of experimental results. The instantaneous pressures at any location on the cylinder were found to be varying between a maximum and minimum during the several shedding cycles. Notably, the pressure excursions were not arbitrarily large, but appear to be bounded above and below by envelops having the same general shape as the distribution of mean pressure.

Many of the numerical calculations in the past were carried out just by comparing the globally varying parameters such as drag coefficient, Strouhal number etc. without paying much attention to the local variation. The complete agreement of local as well as global parameters of the present simulation confirms the efficacy of the chosen numerical scheme. From the C_p distributions, it is seen that the flow past plain cylinders separates at around $\phi = 80^0$ for all the Reynolds numbers covered in this study, as the flow is still in the subcritical regime.

The accuracy of the numerical results depends mainly on the quality of the grid particularly close to the cylinder wall and the time step, apart from the numerical scheme considered. The distance of the first cell $y_{wall}/D \approx 0.004$ is found to be sufficient for flows up to Re = 10^3 while $y_{wall}/D \approx 0.0004$ gives satisfactory results for Re up to 10^5. Near the cylinder wall quadrilateral cells are chosen to resolve the velocity gradients accurately. The time step is set nearly one order less than the time period determined from the anticipated Strouhal number. Further refinement of the grid and time step has not shown significant change in the numerical results of the Reynolds number range covered.

8. Conclusions

Satisfactory results are obtained in the Reynolds number range 200 - 10^5 using a simple approach i.e., unsteady, two-dimensional, second-order upwind scheme and without a turbulence model. The grid close to the cylinder wall is sufficiently refined so as to capture the steep variation of flow parameters accurately. The time step is set nearly one order less than the time period determined from the anticipated Strouhal number. Thereby the computational time required to attain a steady periodic solution is minimized.

When it was tried to predict the flow at higher subcritical Reynolds numbers using lower order turbulent models (k-ε family of models) based on Reynolds averaged Navier-Stokes (RANS) equations, the agreement with experiments was very poor. The inability of the k-ε family of models to predict the bluff body flows might also be due to the assumption of homogeneous and isotropic turbulence. These models often produce excessive eddy viscosity, which induce more damping effects and the separation is delayed, resulting in erroneous pressure distribution. The failure of these models to predict the flow accurately can be explained as follows.

It is found during the simulation that the laminar model works very well up to Re of 10^5. The discrepancy between the present approach and experiments increases with Re beyond this value although it is still lower than the values predicted by the simpler turbulence models. However, laminar models may no longer be applicable well beyond the transition Re.

Although LES and DNS are believed to be capable of giving better results, the high computing cost associated with these and the limited experience on their use have not encouraged their widespread use. They need to be assessed in a careful study about their applicability for a wide range of Reynolds numbers.

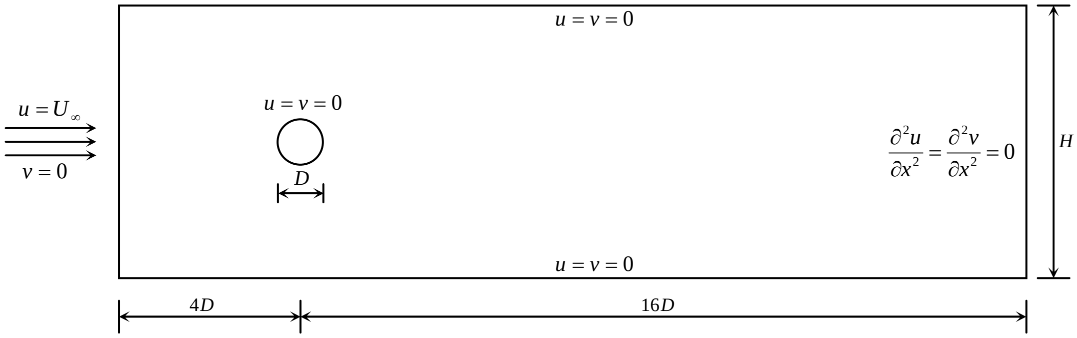


Fig. 1 Schematic diagram of the flow past a circular cylinder

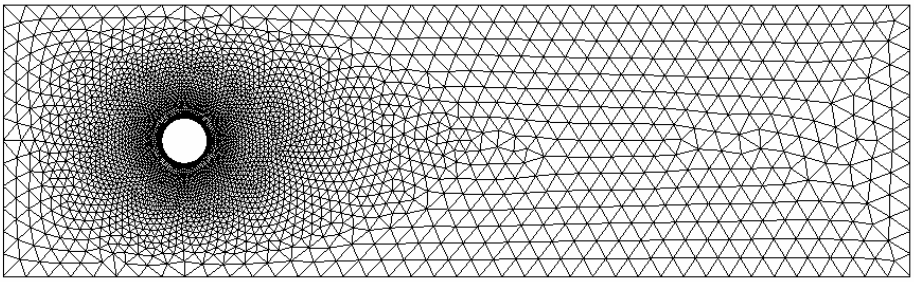


Fig. 2 Computational domain with grid

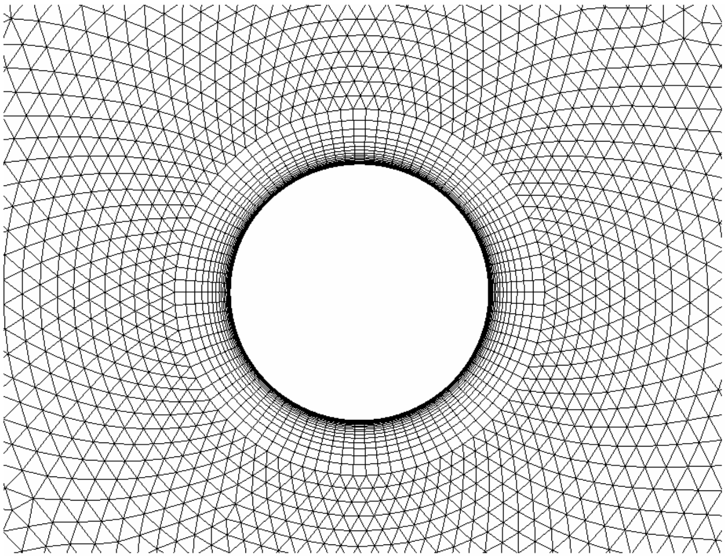
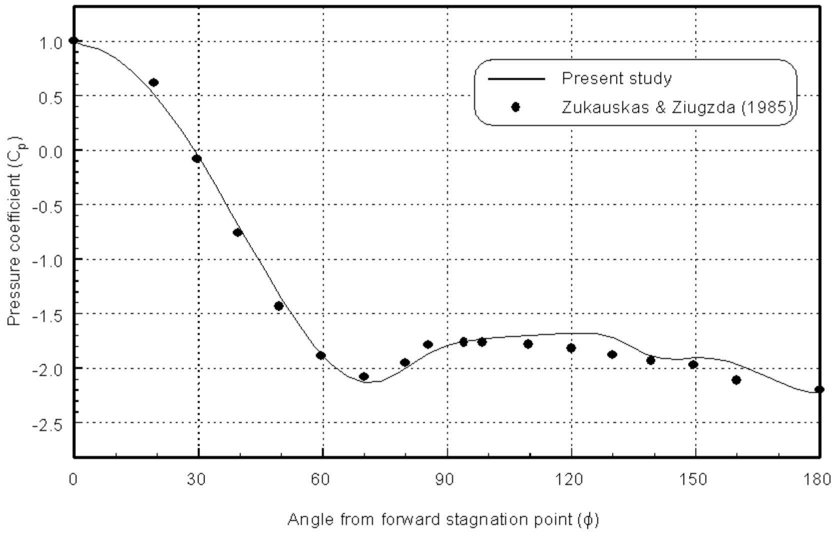
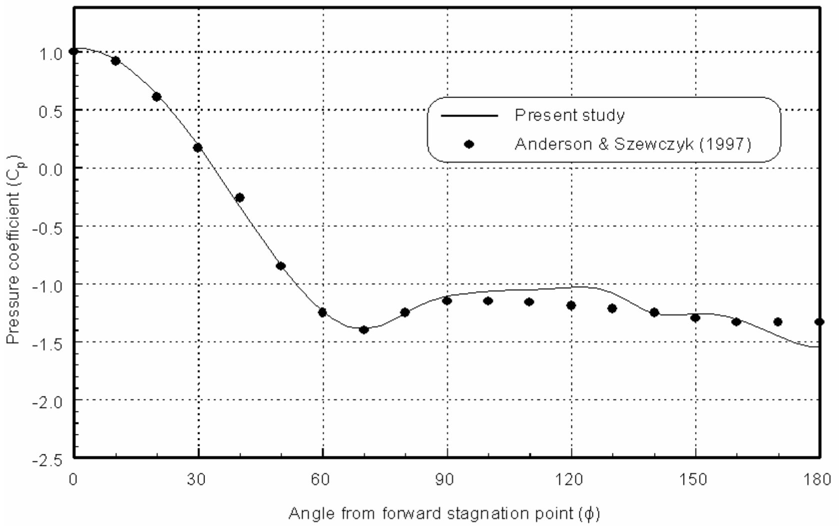


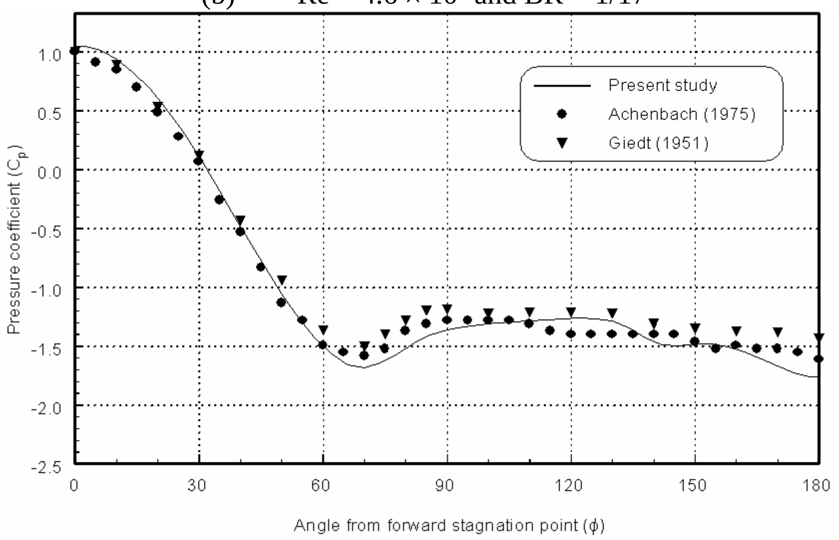
Fig. 3 Close view of the grid



(a) Re = 4.2×10^4 and BR = 1/4



(b) Re = 4.6×10^4 and BR = 1/17



(c) Re = 10^5 and BR = 1/6

Fig. 4 C_p distribution around the cylinder

Nomenclature

BR Blockage ratio, D/H
C_p Pressure coefficient, $(p_\phi - p_\infty)/(½*\rho*U^2_\infty)$
D Cylinder diameter
H Height of the channel
p_ϕ Static pressure at peripheral angle ϕ of the cylinder
p_∞ Free stream static pressure
Re Reynolds number, $\rho*U_\infty*D/\mu$
t Elapsed computational time
u x-directional velocity component
U_∞ Free stream velocity
v y-directional velocity component
ϕ Angle from forward stagnation point on the cylinder

References

E. Achenbach, 'Total and local heat transfer from a smooth circular cylinder in cross flow at high Reynolds number', *International Journal Heat Mass Transfer,* Vol. 18, pp. 1387-1396, 1975.

E.A. Anderson and A.A. Szewczyk, 'Effects of a splitter plate on the near wake of a circular cylinder in 2 and 3 dimensional flow configurations', *Experiments in Fluids*, Vol.23, pp. 161-174, 1997.

M. Braza, P. Chassaing and M. Ha Minh 'Numerical study and physical analysis of the pressure and velocity fields in the near wake of a circular cylinder', *Journal of Fluid Mechanics*, Vol. 165, pp. 79-130, 1986.

M. Breuer, 'Numerical and modeling influences on large eddy simulations for the flow past a circular cylinder', *International Journal of Heat and Fluid Flow*, Vol. 19, pp. 512-521, 1998.

M. Franke, W. Rodi and B. Schonung, 'Numerical calculation of laminar vortex-shedding flow past cylinders', *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 35, pp. 237-257, 1990.

W.H. Giedt, 'Effect of turbulence level of incident air stream on local heat transfer and friction on a cylinder', *Journal of the Aeronautical Sciences*, Vol. 18, pp. 725-730, 1951.

K. Kakuda and N. Tosaka, 'Numerical simulation of high Reynolds number flows by Petrov-Galerkin finite element method', *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 46&47, pp. 339-347, 1993.

M. Kawahara and H. Hirano, 'A finite element method for high Reynolds number viscous fluid flow using two step explicit scheme', *International Journal for Numerical Methods in Fluids*, Vol. 3, pp. 137-163, 1983.

N. Kondo, 'Direct third-order upwind finite element simulation of high Reynolds number flows around a circular cylinder', *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 46&47, pp. 349-356, 1993.

S. Murakami, ‘Comparison of various turbulence models applied to a bluff body’, *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 46&47, pp. 21-36, 1993.

W. Rodi, ‘On the simulation of turbulent flow past bluff bodies’, *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 46&47, pp. 3-19, 1993.

J.S. Son and T.J. Hanratty, ‘Numerical solution for the flow around a cylinder at Reynolds numbers of 40, 200 and 500’, *Journal of Fluid Mechanics*, Vol. 35, 2, pp. 369-386, 1969.

C.C.S. Song and M. Yuan, ‘Simulation of vortex-shedding flow about a circular cylinder at high Reynolds numbers’, *Trans. ASME, Journal of Fluids Engineering*, Vol. 112, pp. 155-163, 1990.

H. Takami and H.B. Keller, ‘Steady two-dimensional viscous flow of an incompressible fluid past a circular cylinder’, *The Physics of Fluids,* Supplement II, pp. 51-56, 1969.

T. Tamura, I. Ohta and K. Kuwahara ‘On the reliability of two-dimensional simulation for unsteady flows around a cylinder-type structure’, *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 35, pp. 275-298, 1990.

A. Thom, ‘The flow past circular cylinders at low speeds’, *Proc. Roy. Soc.*, Vol. A 141, pp. 651, 1933.

D.C. Thoman and A.A. Szewczyk, ‘Time-dependent viscous flow over a circular cylinder’, *The Physics of Fluids,* Supplement II, pp. 76-86, 1969.

A. Zukauskas and J. Ziugzda, *Heat Transfer of a Circular Cylinder in Crossflow*, Hemisphere Publishing Corporation, 1985.