Ahmad Sohankar
Assistant Professor

L. Davidson
Professor

Thermo and Fluid Dynamics,
Chalmers University of Technology,
SE-412 96 Göteborg, Sweden

C. Norberg
Associate Professor
Heat and Power Engineering,
Lund Institute of Technology, Box 118,
SE-221 00 Lund, Sweden

Large Eddy Simulation of Flow Past a Square Cylinder: Comparison of Different Subgrid Scale Models

Large eddy simulation of flow past a rigid prism of a square cross section with one side facing the oncoming flow at $Re=2.2\times10^4$ is performed. An incompressible code is used employing an implicit fractional step method finite volume with second-order accuracy in space and time. Three different subgrid scale models: the Smagorinsky, the standard dynamic, and a dynamic one-equation model, are applied. The influence of finer grid, shorter time step, and larger computational spanwise dimension is investigated. Some global quantities, such as the Strouhal number and the mean and rms values of lift and drag, are computed. A scheme for correcting the global results for blockage effects is presented. By comparison with experiments, the results produced by the dynamic one-equation one give better agreement with experiments than the other two subgrid models. [S0098-2202(00)01001-4]

1 Introduction

The flow around bluff bodies, such as cylinders and prisms, is of relevance to technical problems associated with energy conversion and structural design and arises in many industrial applications and environmental situations. In recent years, researchers' attention has turned to the use of large eddy simulation (LES) for studying turbulent flow around bluff bodies [1–4]. A LES workshop was held in June 1995 in Germany, and the results are published in Rodi et al. [4]. One of the selected test cases at this workshop is the flow around a square cylinder at zero incidence (one side face facing the oncoming flow) for which LDV measurements are reported [5]. The same flow was considered as test case LES2 at the Second ERCOFTAC Workshop on Direct and Large Eddy Simulation in March 1994. Seven groups took part in the LES2 exercise, and the results of this exercise are reported by Voke [6]. The reason for this focus on LES for the study of flow around bluff bodies has to do with poor results when using statistical turbulence models. Most probably this has to do with complicating factors such as a strongly retarded stagnation flow, massive flow separation, streamline curvature, transition from laminar to turbulent flow, recirculation, vortex shedding, and perhaps most important, the existence of inherent three-dimensional flow structures [7,8]. The presence of sharp corners may also be a complicating factor in flow simulations, especially at high Reynolds numbers.

The main objective of the present study was the examination of different subgrid scale (SGS) models of LES of flow around a square cylinder at $\mathrm{Re}=2.2\times10^4$. Another objective was to make a critical evaluation of this selected flow case, in particular on the effects of solid blockage (wall confinement).

2 Configuration and Numerical Details

The flow is described in a Cartesian coordinate system (x,y,z) in which the x axis is aligned with the inlet flow direction, the z axis is parallel with the cylinder axis, and the y axis is perpendicular to both x and z, as shown in Fig. 1. A fixed two-dimensional square cylinder with a side d is exposed to a constant free stream velocity U_∞. An incompressible flow with constant fluid properties is assumed. The Reynolds number is defined as $\mathrm{Re}=U_\infty d/\nu$. All geometrical lengths are scaled with d. Scaling with d also applies for the Strouhal number, $\mathrm{St}=f_S d/U_\infty$, where f_S is the shedding frequency for all forces. In the y direction, the vertical distance between the upper and lower walls H defines the solid blockage of the confined flow (blockage parameter $\beta=1/H$). Velocities are also scaled with U_∞, and physical times with d/U_∞.

Six simulations were performed with different subgrid-scale models: the Smagorinsky model ($C_S=0.1$), the standard dynamic model, and a new dynamic one-equation model. The influence of finer spatial and temporal resolutions, and the size of the spanwise dimension on the results for the dynamic one-equation subgrid scale model, were also investigated. Details on these simulations are provided in Table 1.

An incompressible finite volume code, based on a fractional step technique and employing a nonstaggered grid arrangement, was used. The scheme is implicit in time, and a second-order Crank–Nicolson scheme was used. All terms were discretized using the second-order central differencing scheme, see [9] for greater detail. The time-marching calculations were started with the fluid at rest, and a constant time step Δt was used. The grid distribution was uniform with a constant cell size Δ_u outside a region from the body, which extended two units upstream, downstream, and sideways (in the x and y directions). The distance from the cylinder surface to the nearest grid point defines δ. For all calculations in this study, $\delta\approx0.008$. The hyperbolic tangent function was used for stretching the cell sizes between these limits (δ and Δ_u). A uniform grid with a distance of Δ_z between nodes was used in the spanwise direction (z direction, with spanwise

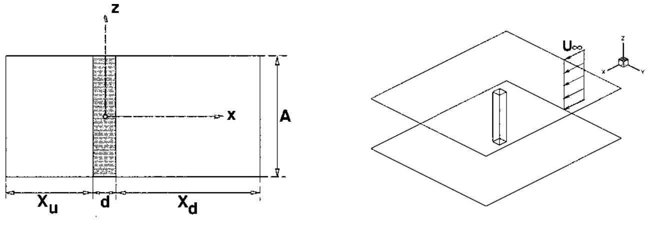

Fig. 1 Flow configuration

Contributed by the Fluids Engineering Division for publication in the JOURNAL OF FLUIDS ENGINEERING. Manuscript received by the Fluids Engineering Division March 15, 1999; revised manuscript received November 15, 1999. Associate Technical Editor: J. Eaton.