Advances in Applied Mathematics and Mechanics
Adv. Appl. Math. Mech., Vol. **2**, No. 4, pp. 451-466

DOI: 10.4208/aamm.09-m0972
August 2010

Large Eddy Simulation of Flow over a Cylinder Using High-Order Spectral Difference Method

Abrar H. Mohammad[1], Z. J. Wang[1,*] and Chunlei Liang[2]

[1] *Department of Aerospace Engineering, Iowa State University, Ames, IA 50011, U.S.A.*

[2] *Department of Aeronautics and Astronautics, Stanford University, Stanford, CA 94305, U.S.A.*

Received 24 July 2009; Accepted (in revised version) 9 October 2009
Available online 28 May 2010

Abstract. Large eddy simulation of the flow over a circular cylinder at Reynolds number $\mathrm{Re_D} = 2580$ has been studied with a high-order unstructured spectral difference method. Grid and polynomial refinement studies were carried out to assess numerical errors. The mean and fluctuating velocity fields in the wake of a circular cylinder were compared with PIV experimental measurements. The numerical results are in excellent agreement with the experimental data for both the mean velocity and Reynolds stresses using the high-order SD scheme. Other wake characteristics such as the recirculation bubble length, vortex formation length and maximum intensity of the velocity fluctuations have also been predicted accurately. The numerical simulations demonstrated the potential of the high-order SD method in accurate large eddy simulation of physically complex problems.

AMS subject classifications: 76F65

Key words: LES, spectral difference, circular cylinder, coherent structures, vortex shedding.

1 Introduction

The flow around bluff bodies at sub-critical Reynolds number is very complex and can involve regions of laminar, transitional and turbulent flows, unsteady separation and reattachment, and the formation of coherent structures, particularly in the wake region of the flow. The understanding of bluff body vortex shedding is of great practical importance and the uniform flow over a circular cylinder is a classical example of bluff body flow. In this paper, we try to show the importance of using high order

*Corresponding author.
URL: `http://www.zjwang.com/`
Email: abrar.hasan@gmail.com (A. H. Mohammad), zjw@iastate.edu (Z. J. Wang), chliang@stanford.edu (C. Liang)

methods to study the numerical and physical aspect of unsteady wake flow involving separation, recirculation, unsteady vortex shedding and large complex flow structures at a sub-critical Reynolds number of $\mathrm{Re_D} = 2580$. At this Reynolds number, we have experimental data to compare with. The near wake structure behind a bluff body plays an important role in the overall vortex formation and shedding processes and determines the magnitude of mean and fluctuating forces exerted on the body. Direct numerical simulations (DNS) of the Navier-Stokes equations, in which all eddy scales have to be captured, is almost impossible for problems with moderately high Reynolds number because of the huge computational requirement in resolving all turbulence scales. Hence a less expensive and accurate method is required. In Reynolds averaged Navier-Stokes (RANS) approach, all eddies are time-averaged over to give equations for variables representing the mean flow. But RANS has proved to be generally inadequate in predicting the effects of turbulent separating and reattaching flows, because the large eddies responsible for the primary transport are geometry dependent. For any turbulent flow, the largest scale is of the order of the domain size and the small scales are related to the dissipative eddies where the viscous effects become predominant. Large eddy simulation (LES) is a method where the three-dimensional and unsteady motion of the large eddies is computed explicitly and the non-linear interactions with the smaller eddies, which are assumed to be isotropic and universal, are modeled. LES is an active area of research and the numerical simulation of complex flows is essential in the development of the method as a tool to predict flows of engineering interest.

In this paper, implicit LES computations were performed without any sub-grid scale model in order to investigate the effectiveness of the spectral difference method. These simulations were deliberately not called direct numerical simulations because they did not comply with the resolution requirements of DNS. Turbulent flow past a circular cylinder has been the subject of a large number of experimental and numerical investigations. Examples can be found in the review papers by Williamson and Govardhan [44] and Sarpkaya [33]. In recent years a good understanding of the physics of flow at low Reynolds number of below a few hundred, has been obtained. But at higher Reynolds number, still subcritical though, considerably less is known. A comprehensive review of the flow characteristics for a wide range of Reynolds numbers was studied by Williamson [43]. In addition, a number of simulations at various Reynolds numbers, mostly LES, have been carried out, such as Travin et al. [37], Breuer [2, 3], Liang and Papadakis [22] and Catalano et al. [4]. The cylinder flow at Reynolds number $\mathrm{Re_D} = 3900$ has become a common test case for LES primarily because of the availability of the experimental results of Lourenco and Shih [20] and Ong and Wallace [31]. The calculations were performed on structured [1–3, 18, 28] and unstructured meshes [10, 12, 22, 26]. Beaudan and Moin [1], Breuer [2, 3], Mittal and Moin [28], Kravchenko and Moin [18] were among the first to perform LES studies at $\mathrm{Re_D} = 3900$. Motivated by the direct simulation results of Rai and Moin [32], Beaudan and Moin [1] used high-order upwind-biased schemes for the numerical simulations of the compressible Navier-Stokes equations. The profiles of mean velocity