Commun. Comput. Phys.
doi: 10.4208/cicp.050513.270513a

Vol. **15**, No. 2, pp. 388-421
February 2014

Constrained Large-Eddy Simulation of Compressible Flow Past a Circular Cylinder

Renkai Hong[1], Zhenhua Xia[1], Yipeng Shi[1,2,*], Zuoli Xiao[1,2,*] and Shiyi Chen[1,2,*]

[1] *State Key Laboratory for Turbulence and Complex Systems, College of Engineering, Peking University, Beijing 100871, P.R. China.*
[2] *Center for Applied Physics and Technology, College of Engineering, Peking University, Beijing 100871, P.R. China*

Received 5 May 2013; Accepted 27 May 2013

Abstract. Compressible flow past a circular cylinder at an inflow Reynolds number of 2×10^5 is numerically investigated by using a constrained large-eddy simulation (CLES) technique. Numerical simulation with adiabatic wall boundary condition and at a free-stream Mach number of 0.75 is conducted to validate and verify the performance of the present CLES method in predicting separated flows. Some typical and characteristic physical quantities, such as the drag coefficient, the root-mean-square lift fluctuations, the Strouhal number, the pressure and skin friction distributions around the cylinder, *etc*. are calculated and compared with previously reported experimental data, finer-grid large-eddy simulation (LES) data and those obtained in the present LES and detached-eddy simulation (DES) on coarse grids. It turns out that CLES is superior to DES in predicting such separated flow and that CLES can mimic the intricate shock wave dynamics quite well. Then, the effects of Mach number on the flow patterns and parameters such as the pressure, skin friction and drag coefficients, and the cylinder surface temperature are studied, with Mach number varying from 0.1 to 0.95. Non-monotonic behaviors of the pressure and skin friction distributions are observed with increasing Mach number and the minimum mean separation angle occurs at a subcritical Mach number of between 0.3 and 0.5. Additionally, the wall temperature effects on the thermodynamic and aerodynamic quantities are explored in a series of simulations using isothermal wall boundary conditions at three different wall temperatures. It is found that the flow separates earlier from the cylinder surface with a longer recirculation length in the wake and a higher pressure coefficient at the rear stagnation point for higher wall temperature. Moreover, the influences of different thermal wall boundary conditions on the flow field are gradually magnified from the front stagnation point to the rear stagnation point. It is inferred that the CLES approach in its current version is

---

[*] Corresponding author. *Email addresses:* `hongrenkai@163.com` (R. Hong), `xiazh@pku.edu.cn` (Z. Xia), `syp@mech.pku.edu.cn` (Y. Shi), `z.xiao@pku.edu.cn` (Z. Xiao), `syc@pku.edu.cn` (S. Chen)

a useful and effective tool for simulating wall-bounded compressible turbulent flows with massive separations.

AMS subject classifications: 76F50, 76F65, 76H99, 76G25

Key words: Compressible flow, constrained large-eddy simulation, circular cylinder, separated flow.

1 Introduction

Flow around a circular cylinder is of great importance, either in practical engineering problems such as wind past a cooling tower, lateral flow past an aircraft body, or in fundamental research problems. As the cylinder surface is smooth and curved, a sensitive boundary layer forms and will separate from the surface if the inflow Reynolds number is high enough. The transition of flow to turbulence may take place either in the boundary layer (*i.e.*, turbulent separation) or in the separated shear layer (*i.e.*, laminar separation). The interactions among the three shear layers, say, the boundary layer, the separated free shear layer and the wake, result in complex physical phenomena in incompressible case [1]. The first experimental study on this problem was carried out by Bénard [2] at the beginning of the twentieth century. Then, von Kármán [3] and Thom [4] investigated the same problem theoretically and numerically, respectively. These seminal works have sparked increasing interests and challenges in understanding the intricate dynamics of flow past a circular cylinder. The readers are referred to the review articles [5–8] and textbooks [9, 10] for details. In fact, the geometrical simplicity and the availability of numerical and experimental data of flow past a circular cylinder have suggested that it be a benchmark model to validate a numerical solver or a turbulence model both in incompressible cases [11–18] and in compressible cases [19–21].

In compressible flow past a circular cylinder, especially at transonic inflow conditions, the problem becomes more complicated due to the three mutually coupled fundamental processes (*i.e.*, compressing, shearing, and thermal). Related topics include compressible boundary layer instability, shock wave/boundary layer interactions, shock wave/wake interactions, *etc*. Macha systematically examined the pressure distributions and the drag coefficients around a circular cylinder in a wind tunnel at Reynolds and Mach numbers from 0.1×10^6 to 1.0×10^6 and from 0.6 to 1.2, respectively [22]. As Mach number increases, it is shown that the pressure drag keeps increasing before Mach 0.7, remains constant or decreases slightly from Mach 0.7 to Mach 0.9, increases dramatically near Mach 1.0 and decreases monotonically when Mach number is larger than unit. It is argued that the leveling-off of the drag variation prior to the major drag crisis near Mach 1.0 be attributed to the compressibility effects. Murthy and Rose [23] also conducted a series of wind tunnel experiments on air flow past a circular cylinder with Mach and Reynolds numbers varying from 0.25 to 1.2 and from 0.03×10^6 to 0.5×10^6, respectively. The transonic drag rise phenomenon was also observed. Meanwhile, it is found that detectable