Computers and Structures 87 (2009) 812–827

Contents lists available at ScienceDirect

Computers and Structures

journal homepage: www.elsevier.com/locate/compstruc

High-order accurate simulation of low-Mach laminar flow past two side-by-side cylinders using spectral difference method

Chunlei Liang *, Sachin Premasuthan, Antony Jameson

Aeronautics and Astronautics, Stanford University, Durand 005, Stanford, CA 94305, United States

ARTICLE INFO

Article history:
Received 10 December 2008
Accepted 22 December 2008
Available online 27 February 2009

Keywords:
Spectral difference method
Curved wall boundary
Unstructured grid

ABSTRACT

This paper reports development of a two-dimensional solver for compressible viscous flow using spectral difference (SD) method and its applications on simulating laminar flow past two side-by-side cylinders at various spacings. The high-order spectral difference solver is based on unstructured quadrilateral grids. High-order curved wall boundary representation is developed for cylinders. Nine different spacings (center-to-center distance/diameter $s = 1.1$, 1.4, 1.5, 1.7, 2, 2.5, 3, 3.4 and 4) are investigated. The simulation results are compared to experimental results and other numerical results. As s increases, single bluff-body, flip-flopping, anti-symmetric and symmetric wake patterns are predicted.

Published by Elsevier Ltd.

1. Introduction

1.1. Flow past two side-by-side cylinders

Investigations of the fluid flow and vortex dynamics about simple configurations of two cylinders help our understanding of the flows around more complex and larger-scale structures, for instance the flow around tube banks employed in process industries [14] and especially in the power generation and oil industry as well as flow around neighboring buildings and river flow vegetation, etc. Other applications are also related to two cylinders such as hollow fiber arrays with many applications in absorption, extraction and ultra-filtration [12] or paper machine forming fabrics [9]. In the latter examples, the flows are laminar with Reynolds number in the range of 150–200.

Zdravkovich [34,35] has reviewed the problem of mutual interference between pairs of cylinders in a steady flow. Much attention was paid to the side-by-side and inline arrangements of the cylinder pair. Williamson [31] suggested that a spacing between two side-by-side cylinders with the ratio of distance between cylinder centers to the diameter (s) in the range of 2–6 produces vortex-shedding synchronization. The resulting wake configuration will be either two parallel streets in antiphase mode or a binary-vortex street mode which consists of a street being composed of pairs of like-signed vortices rotating around one another with Reynolds number in the range of 100–200. Experimental results were also obtained by Zhou et al. [36] at relatively low Reynolds numbers (150–450). They suggested that the flow pattern is very much independent of Reynolds number of this range. At $s = 3$, they observed the anti-phase flow patterns for all Reynolds numbers using more advanced flow visualization methods. Chang and Song [3] made an early investigation of laminar flow past two side-by-side cylinders using a blending technique of finite-element method and finite-difference method. Recently, numerical simulations have been performed for incompressible laminar flow past two side-by-side cylinders by various methods. For instance, Meneghini et al. [18] used a finite-element unstructured method, Kang [10] and Lee et al. [11] employed a finite-volume structured method with immersed boundary technique and Ding et al. [7] developed a mesh-free finite-difference method and studied this particular configuration.

The above studies mentioned are all about incompressible flows. The simulation codes commonly attained at best second-order accuracy in space. Furthermore, all the above discussed numerical simulations employed only piecewise linear wall boundary conditions or some kind of interpolation schemes to satisfy no-slip condition for immersed boundary method. The present simulation uses a recently developed Spectral Difference high-order unstructured method to simulate a low-Mach number compressible laminar flow past two side-by-side cylinders. A cubic spline curve fitting routine is programmed into our solver and it allows an automatic construction of a cubic curved wall boundary condition for each cylinder.

1.2. Spectral difference method

Until recently, compressible flow computations on unstructured meshes have generally been dominated by schemes restricted to second order accuracy. However, the need for highly

* Corresponding author. Tel.: +1 650 724 5479.
E-mail address: chliang@stanford.edu (C. Liang).

0045-7949/$ - see front matter Published by Elsevier Ltd.
doi:10.1016/j.compstruc.2008.12.016

accurate methods in applications such as large eddy simulation, direct numerical simulation, computational aero-acoustics etc., has seen the development of higher order schemes for unstructured meshes such as the discontinuous Galerkin (DG) method [4,5,1], spectral volume (SV) method [29,17,30] and spectral difference (SD) method [16,28,13]. The SD method is a newly developed efficient high-order approach based on differential form of the governing equation. It was originally proposed by Liu et al. [16] and developed for wave equations in their paper on triangular grids. Wang et al. [28] extended it to 2D Euler equations on triangular grids and Liang et al. [13] improved the convergence of the method using implicit LU-SGS and p-multigrid schemes. Recently, Sun et al. [27] further developed it for three-dimensional Navier–Stokes equations on hexahedral unstructured meshes. Mohammad et al. [21] investigated flow past a circular cylinder at subcritical Reynolds number using the SD method. The SD method combines elements from finite-volume and finite-difference techniques. Similar to the discontinuous Galerkin (DG) and spectral volume (SV) methods, the SD scheme achieves high-order accuracy by locally approximating the solutions as a high degree polynomial inside each cell. However, being based on the differential form of the equations, its formulation is simpler than that of the DG and SV methods as no test function or surface integral is involved. Conservation properties are still maintained by a judicious placement of the nodes at quadrature points of the chosen simplex.

This paper presents development of a new in-house two-dimensional high-order SD code for viscous compressible flow. The formulations are similar to the ones used by Sun et al. [27]. Previous numerical studies on two side-by-side cylinders have not concluded the effect of tube spacings on flow separation points, wake flow pattern and force coefficients. The SD method is employed in this paper to study the unsteady laminar flow past a pair of side-by-side cylinders with nine different spacings (center-to-center distance/diameter s = 1.1, 1.4, 1.5, 1.7, 2, 2.5, 3, 3.4 and 4). We aim to see the effect of the spacings on flow pattern, separation points and flow exerted forces.

The paper is organized as follows. Section 2 describes the numerical approach and solution methods. In order to validate the spatial accuracy of the code, Section 3 presents two cases with analytical solutions and simulation results obtained by the SD method in addition to a simulation of flow past an isolated cylinder with detailed comparisons to other results. Section 4 reports the simulation results obtained for laminar viscous flows past two side-by-side cylinders. Finally, Section 5 summarizes the main findings of this work.

2. Numerical formulation

The formulation of the equations is similar to the formulation of Sun et al. [27] for unstructured hexahedral grids.

Consider the unsteady compressible 2D Navier–Stokes equations in conservative form

$$\frac{\partial Q}{\partial t}+\frac{\partial F}{\partial x}+\frac{\partial G}{\partial y}=0 \tag{1}$$

where Q is the vector of conserved variables; F and G are the total fluxes including both inviscid and viscous flux vectors. To achieve an efficient implementation, all elements in the physical domain (x, y) are transformed into a standard square element $(0 \leqslant \xi \leqslant 1, 0 \leqslant \eta \leqslant 1)$ as shown in Fig. 1. The transformation can be written as

$$\begin{pmatrix} x \\ y \end{pmatrix} = \sum_{i=1}^{K} M_i(\xi,\eta)\begin{pmatrix} x_i \\ y_i \end{pmatrix} \tag{2}$$

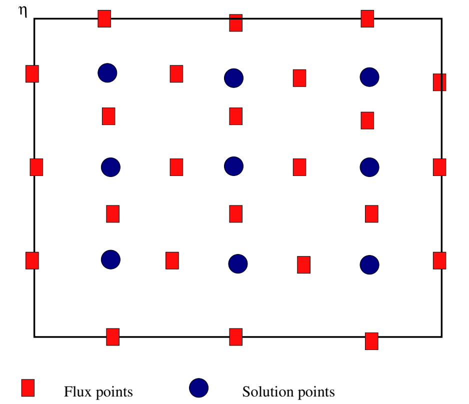

Fig. 1. Distribution of flux and solution points for the third order SD scheme.

where K is the total number of points used to define the physical element, (x_i, y_i) are the cartesian coordinates of those points, and $M_i(\xi,\eta)$ are the shape functions. For elements with straight edges, K is equal to 4. For elements lying on curved boundaries, 8 points (four mid-edge and four corner points) can define a quadratic representation and 12 points can determine a third-order cubic representation. The metrics and the Jacobian of the transformation can be computed for each element. The Jacobian can be expressed as follows:

$$J = \begin{pmatrix} x_\xi & x_\eta \\ y_\xi & y_\eta \end{pmatrix} \tag{3}$$

The governing equations in the physical domain are then transferred into the computational domain, and the transformed equations take the following form:

$$\frac{\partial \tilde{Q}}{\partial t}+\frac{\partial \tilde{F}}{\partial \xi}+\frac{\partial \tilde{G}}{\partial \eta}=0 \tag{4}$$

where $\tilde{Q} = |J| \cdot Q$ and

$$\begin{pmatrix} \tilde{F} \\ \tilde{G} \end{pmatrix} = |J|J^{-1}\begin{pmatrix} F \\ G \end{pmatrix} \tag{5}$$

In the standard element, two sets of points are defined, namely the solution points and the flux points as illustrated in Fig. 1.

In order to construct a degree $(N-1)$ polynomial in each coordinate direction, solutions at N points are required. The solution points in 1D are chosen to be the Gauss points defined by

$$X_s = \frac{1}{2}\left[1-\cos\left(\frac{2s-1}{2N}\cdot\pi\right)\right], \quad s = 1,2,\ldots,N \tag{6}$$

The flux points are selected to be the Gauss–Lobatto points given by

$$X_{s+1/2} = \frac{1}{2}\left[1-\cos\left(\frac{s}{N}\cdot\pi\right)\right], \quad s = 0,1,\ldots,N \tag{7}$$

Using the solutions at N solution points, a degree $(N-1)$ polynomial can be built using the following Lagrange basis:

$$h_i(X) = \prod_{s=0,s\neq i}^{N}\left(\frac{X-X_s}{X_i-X_s}\right) \tag{8}$$

Similarly, using the fluxes at $(N+1)$ flux points, a degree N polynomial can be built for the flux using a similar Lagrange basis

$$l_{i+1/2}(X) = \prod_{s=0,s\neq i}^{N}\left(\frac{X-X_{s+1/2}}{X_{i+1/2}-X_{s+1/2}}\right) \tag{9}$$