Journal of Computational Physics 301 (2015) 425–455

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

Spectrally-accurate algorithm for the analysis of flows in two-dimensional vibrating channels

S. Zandi, A. Mohammadi *, J.M. Floryan

Department of Mechanical and Materials Engineering, The University of Western Ontario, London, Ontario, N6A 5B9, Canada

ARTICLE INFO

Article history:
Received 25 February 2015
Received in revised form 23 July 2015
Accepted 17 August 2015
Available online 24 August 2015

Keywords:
Spectral discretization
Vibrating walls
Immersed boundary conditions method

ABSTRACT

A spectral algorithm based on the immersed boundary conditions (IBC) concept has been developed for the analysis of flows in channels bounded by vibrating walls. The vibrations take the form of travelling waves of arbitrary profile. The algorithm uses a fixed computational domain with the flow domain immersed in its interior. Boundary conditions enter the algorithm in the form of constraints. The spatial discretization uses a Fourier expansion in the stream-wise direction and a Chebyshev expansion in the wall-normal direction. Use of the Galileo transformation converts the unsteady problem into a steady one. An efficient solver which takes advantage of the structure of the coefficient matrix has been used. It is demonstrated that the method can be extended to more extreme geometries using the overdetermined formulation. Various tests confirm the spectral accuracy of the algorithm.

1. Introduction

The determination of flows in channels with vibrating walls requires solving a moving boundary problem. This class of problems has been of interest in many application areas including surface waves, interfacial problems, phase change problems, flow induced vibrations, peristaltic and pulsatile flows in the esophagus and flows through the vasculatures due to cardiac actions, to name just a few. The available algorithms can be classified either as Lagrangian or Eulerian [1]. Mixed methods involving combinations of Lagrangian and Eulerian techniques have also been pursued [1]. Each fluid element is followed individually in the Lagrangian algorithms, resulting in a need for a coordinate system that moves with the fluid. Mesh tangling leads to significant restrictions on the overall applicability of these methods [1]. The Eulerian algorithms rely on coordinate systems that are stationary in a laboratory frame of reference or may move in a prescribed manner. Such algorithms can be divided into fixed grid methods, adaptive grid methods and various mapping methods.

In the fixed grid methods, the grid is fixed in the solution domain and the locations of the moving boundaries are tracked using either surface [1,2] or volume tracking procedures [1,3]. Surface tracking relies on a set of points whose motion is tracked during the solution process, allowing precise identification of the boundary locations; these boundaries are represented as a set of interpolated curves [3,4]. The volume tracking algorithms, on the other hand, work by reconstructing the boundary whenever necessary instead of storing the boundary locations. The presence of a convenient marker within a computational cell and its quantity form the basis of the various reconstruction methodologies. Different versions of volume tracking algorithms exist, e.g. VOF (Volume of Fluid) [5], MAC (Marker and Cell) [6] and Level Set [7,8] methods. These

* Corresponding author.
E-mail address: amoham69@alumni.uwo.ca (A. Mohammadi).

http://dx.doi.org/10.1016/j.jcp.2015.08.025
0021-9991/

methods are based on standard spatial discretization schemes with low-order spatial accuracy consistent with the diffused boundary locations resulting from the tracking procedures.

The adaptive grid methods use numerical mappings to adjust the grids at each time step so that one of the grid lines always overlaps with the boundary location. The computational costs are very high due to grid reconstruction at each time step, e.g. the grid construction consumed about 75% of the computational cost for the problem discussed in [9]. The spatial discretization technique has a smaller effect on the overall computational costs. The need for high solution accuracy leads to numerous challenges as the total error has contributions from the grid generation as well as from the spatial and temporal discretizations of the field equations. The use of mappings based on the Schwarz–Christoffel transformation provides access to higher accuracy at a reasonable cost as one only needs to determine mapping parameters and these parameters can be determined with near-spectral accuracy [10–12]. An analytical mapping of the irregular physical domain to a rectangular computational domain can help improve the accuracy at the cost of increased complexity of the field equations [13,14]. However, such mappings are available only for a limited class of geometries [1] and reconstruction of the coefficient matrix during each time step can add to the overall computational cost by a substantial margin [14–20].

Immersed or fictitious boundaries represent a new concept with the potential to increase the accuracy while maintaining the computational efficiency. This concept is due to Peskin [21] and has been developed in the context of cardiac dynamics; see [22,23] for reviews. The common limitation is the spatial accuracy, as most of these methods are based on either low-order finite-difference, finite-volume or finite-element techniques [22–26]. The second, less known limitation is the use of the local fictitious forces required to enforce the no-slip and no-penetration conditions. These forces locally affect the flow physics and this may lead to incorrect estimates of derivatives of flow quantities, i.e. misrepresentation of the local wall shear. This problem is likely to be more pronounced in the case of methods with high spatial accuracy.

Spectral methods provide the lowest error for the spatial discretization but are generally limited to solution domains with regular geometries. The first spectrally accurate implementation of the immersed boundary concept is given in [27]. We shall refer to this method as the immersed boundary conditions (IBC) method in the rest of this presentation. The IBC method relies on a purely formal construction of boundary constraints in order to generate the required closing relations. The spatial discretization relies on Fourier and Chebyshev expansions in the stream-wise and wall-normal directions, respectively, and thus provides the ability to attain machine level accuracy. The method could be viewed as gridless as it uses global basis functions which span the complete solution domain. The construction of boundary constraints relies on the representation of the physical boundaries in the spectral space and nullifying the relevant Fourier modes. The method involves two types of Fourier expansions, one for the field equations and one for the boundary relations and, thus, the rate of convergence of both expansions determines the limits of its applicability. The programming effort associated with accounting for changes in geometry is reduced to the specification of a set of Fourier coefficients which need to be provided as an input. The additional attractiveness of the IBC method is associated with the precise mathematical formalism, high accuracy and sharp identification of the location of time-dependent physical boundaries. The method has been extended to two-dimensional unsteady problems [28], moving boundary problems involving Laplace [29] and biharmonic [30] operators, the complete Navier–Stokes system [31], to operators involving different classes of non-Newtonian fluids [32,33], to three-dimensional operators [34,35] as well as to operators expressed in cylindrical coordinate systems [36]. Its accuracy has been improved through the use of the overdetermined formulation [37]. The efficiency has been increased by an order of magnitude through the development of specialized solvers which account for the special structure of the coefficient matrix [38,39]. The method has been used to identify the laminar drag-reducing grooves [16–20] and to study the effects of various grooves on the flow stability [40–46].

This work is focused on the development of an efficient algorithm suitable for the analysis of changes in the pressure gradient required to drive a specified flow rate through a vibrating channel. Vibrations in the form of travelling waves, such as those found in peristaltic pumping, are of primary interest. The identification of the most effective forms of such waves is of interest. Section 2 provides the problem formulation. Section 3 describes the form of the field equations suitable for the numerical solution. Section 4 discusses the discretization of the field equations. Section 5 provides a description of the proper construction of the boundary constraints. Section 6 discusses the iterative solution procedure. Section 7 describes the linear solver used in the solution. Section 8 provides descriptions of various numerical tests which demonstrate the spectral accuracy of the algorithm. Section 9 describes improvements resulting from the overdetermined formulation. Section 10 describes the validation of the algorithm. Section 11 provides a short summary of the main conclusions.

2. Problem formulation

Consider steady, two-dimensional flow of a fluid confined in a channel bounded by two parallel walls extending to $\pm\infty$ in the X-direction and placed at a distance $2h$ apart as shown in Fig. 1. The flow is driven in the positive X-direction by a pressure gradient resulting in the velocity and pressure fields, and the flow rate of the form

$$\boldsymbol{v_0}(X,Y) = \left(1-Y^2, 0\right), \qquad p_0(X,Y) = -2X/Re, \qquad \Psi_0 = Y - \frac{Y^3}{3} + \frac{2}{3}, \qquad Q_0 = \frac{4}{3} \tag{2.1}$$

where $\boldsymbol{v_0} = (u_0, v_0)$ denotes the velocity vector scaled with the maximum of the X-velocity u_{max}, p_0 stands for the pressure scaled with ρu_{max}^2 where ρ stands for the density, Ψ_0 stands for the stream function, Q_0 denotes the flow rate, the Reynolds number is defined as $Re = u_{max}h/\nu$ where ν stands for the kinematic viscosity, and h has been used as the length scale.