International Journal of Heat and Mass Transfer 55 (2012) 6916–6926

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Mass Transfer

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

An embedding finite element analysis of heat transfer on the surface of circular cylinders in flow

D.C. Lo [a,*], Dong-Taur Su [b]

[a] Institute of Maritime Information and Technology, National Kaohsiung Marine University, Kaohsiung 805, Taiwan
[b] Department of Shipping Technology, National Kaohsiung Marine University, Kaohsiung, Taiwan

ARTICLE INFO

Article history:
Received 30 March 2012
Received in revised form 26 June 2012
Accepted 2 July 2012
Available online 30 July 2012

Keywords:
Finite element analysis
Navier–Stokes equations
Convective heat transfer

ABSTRACT

The present study establishes an embedding finite element method appropriate for solving primitive variable forms of the Navier–Stokes equations and energy equation in a complex physical domain. The stationary solid obstacle in the flow domain is embedded in a non-uniform Cartesian grid and the governing equations are calculated through a finite element formulation. A compact interpolating scheme near the immersed boundaries is used to ensure the accuracy of the solution in the cut cells. We have developed a numerical algorithm based on the operator splitting technique, balance tensor diffusivity (BTD), Runge–Kutta time-stepping method, and a bi-conjugate gradient iterative solver. Three numerical examples are chosen to test the accuracy and flexibility of the proposed scheme. Simulation of flow past a stationary circular cylinder is conducted to validate the accuracy of the present method for solving heat transfer problems. Flow over circular cylinders in a tandem arrangement and a staggered tube bank with convective heat transfer is computed to demonstrate the model's ability to handle complex geometries.

1. Introduction

This paper presents an embedding finite element analysis of heat transfer problems with fluid structure interactions. In the governing equations used to illustrate incompressible viscous flows with force-convection problems, the effect of buoyancy and compressibility are neglected. Thus, the momentum and continuity equations and energy equation are decoupled and only one-way interaction from the governing equations is considered. This is because the Navier–Stokes equations are coupled through the velocity, but the pressure does not appear in the continuity equation. To resolve this, we adopt a decoupled numerical algorithm based on the projection method to overcome the difficulties of coupling between the continuity and momentum equations. The projection method has been pioneered by Chorin [1] to represent the Navier–Stokes equations for the solution of multi-dimensional flow problems. This projection method, also called the fractional step method, has been studied over the past years by many investigators, who have examined this method by using approximations of the advection term and the scheme used for the time discretization [2–6]. The main advantage of the projection is that the incompressibility conditions yield an approximate velocity field, which was made divergence-free from an orthogonal projection of the apparent velocity field. The method has been verified to simulate complex flows and accordingly is widely used in the field of computational fluid dynamics.

In this study, we have taken advantage of the operator splitting technique, balance tensor diffusivity (BTD), and Runge–Kutta time-stepping method based on the projection method for solving 2D viscous flow problems with heat transfer. To reduce computational effort, the system of the equations is solved simultaneously by Jacobi iteration using the mass lumping technique, thus avoiding the formulation of global matrices. Only the pressure Poisson equation is solved by using a bi-conjugate gradient iterative scheme [7], and this is done by storing the non-zero values during calculations. A bi-conjugate gradient iterative scheme is adopted to solve the simultaneous equations to obtain the solution at the global nodes. The implementation of the present iterative solution procedure to solve CFD problems on a personal computer is straightforward because the scheme has been efficiently implemented to accommodate the requirement of small computer memory to store the non-zero entries of the matrices.

Cartesian meshes have commonly been used for solving problems with irregular geometry through the body-fitted or unstructured grid methods. But the computational savings of body-fitted or unstructured finite element meshes become a huge challenge when solving problems with a moving boundary. Our method, in contrast, uses the generation of a structural Cartesian grid rather than that required by body-fitted or unstructured finite element meshes. On the other hand, the underlying Cartesian mesh is used as a powerful tool for saving computational time. In the past

* Corresponding author.
E-mail address: loderg@webmail.nkmu.edu.tw (D.C. Lo).

0017-9310/$ - see front matter
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2012.07.005

several decades the immersed boundary method has become more popular, following its introduction by Peskin [8], due to its simplicity and flexibility in solving moving complex boundary problems with less computational cost and memory requirements than other methods.

Mittal and Iaccarino [9] review important claims that the IB method to simulate incompressible viscous flows can be classified into two major categories, namely continuous forcing [8,10–12] and discrete forcing [13–15]. An early development in the history of the continuous forcing IB method was the note by Peskin [8] that the Navier–Stokes should add a forcing term before the discretizations. A number of studies [10–12,16,17] have investigated for the modifications and improvements of this method since Peskin's paper [8]. In the continuous forcing IB method, the boundaries are modeled as pure force-generators for the simulation of the Navier–Stokes equations related to fixed or moving obstacles interacting with a fluid. It has been argued that the forcing term appearing in the discrete forcing method can be either explicitly or implicitly applied to the discretization of Navier–Stokes [13–15]. In this paper, we present numerical solution algorithm to solve viscous flows with heat transfer in combination with the finite element method and an embedding method based on an interpolation scheme to handle the immersed boundary in the complex geometry. The key issue in using Cartesian grid methods is the imposition of boundary conditions at the immersed boundary. To resolve this, we adopt a compact interpolation scheme near the immersed boundaries that allows us to retain second-order accuracy of the solver. In the presence of the flow past several cylinders with an arbitrary arrangement, the values of the flow variables at the immersed cells are uploaded using a local reconstruction scheme involving the virtual point and the projection node at the immersed boundary. A second order interpolation scheme is used to solve the values of the variables at the immersed cells. Thus the use of finite element method has enabled us to solve a mesh generation problem of great complexity. Finite element method is found to be an effective tool to compute the flow variables due to its good flexibility and stability. We should treat carefully with the immersed cells and boundary condition to ensure a conservative solution. Thus the present numerical procedure exploits the advantages of both the interpolating scheme at the immersed cells and the finite element method in the computational domain.

Numerous experimental and numerical studies have conducted heat-transfer over a stationary circular cylinder. Churchill and Bernstein [18], Eckert and Soehngen [19], and Roshko [20] provided extensive discussions of the experimental results from these studies. Using numerical methods, Momose and Kimoto [21] and Bharti et al. [22] analyzed out the heat transfer over a stationary cylinder at a Reynolds-number range similar to the present study. Since the IB method is concerned, Yoon et al. [23] investigated IB finite volume method for 2D laminar fluid flow and heat transfer past a circular cylinder near a moving wall. Recently, Kim et al. [24] studied the natural convection induced by the temperature difference between a cold outer square enclosure and a hot inner circular cylinder using an IB method. Feng and Michaelides [25] applied the IB method with a difference method to solve the thermal convection in particulate flows. Wang et al. [17] presented a direct source scheme to the simulations of natural convection between concentric cylinders, and analyzed the flow past a stationary circular cylinder to validate the accuracy of the present method for solving heat transfer problems.

The flow past two cylinders is a significant topic of fluid–structure interaction. The arrangement of the cylinders vs. the flow direction of the free stream is used to simulate the hydrodynamic interaction between two cylinders. The arrangement of the cylinders with respect to the free stream flow direction can be classified into three major categories, namely tandem – the free stream flow direction is parallel with the line of the centers of the cylinders, transverse – the free stream flow direction is perpendicular to the line of the cylinders centers, staggered. The flow past a pair of cylinders in tandem arrangement have been thoroughly investigated numerically in [26–32]. To our knowledge, there is less publication in respect of the isothermal flow past two cylinders in tandem [33–35]. Buyruk [33] used numerical method to study the force convection heat transfer for tandem, in-line and staggered cylinders configurations. Juncu [34,35] presents a computational study of the steady and viscous flow around two tandem cylinders. He reported results for force convection heat transfer around two tandem circular cylinders at low Reynolds numbers using a compact finite difference method, Reynolds number varying from 1 to 30 and fluid phase Prandtl number equal to 0.1, 1, 10 and 100. The present study is emphasized to the analysis of forced convection heat transfer from two tandem cylinders in an unsteady and viscous flow.

The contents of this paper are organized as follows. Section 2 revisits the governing equations for 2D viscous flow with forced convection. Section 3 delineates the finite element scheme based on projection method to discretize the governing Navier–Stokes equations in primitive variables form. In Section 4, numerical results for three examples are examined: the effect of heat transfer phenomena for flow past a circular cylinder, two cylinders in a tandem arrangement, and flow past a staggered tube bank which is composed by 77 circular cylinders with heat transfer. Concluding remarks are presented in Section 5.

2. Governing equations

The governing equations for viscous flow can be expressed by means of the non-dimensional form of the Navier–Stokes equations with forced convection in primitive variables form as:

Continuity equation

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{1}$$

Momentum equations

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{\partial p}{\partial x}+\frac{1}{\mathrm{Re}}\nabla^2 u \tag{2}$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{\partial p}{\partial x}+\frac{1}{\mathrm{Re}}\nabla^2 v \tag{3}$$

Energy equation

$$\frac{\partial T}{\partial t}+u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}=\frac{1}{Pe}\nabla^2 T \tag{4}$$

for a Cartesian coordinate frame in which x–y represents the horizontal and the vertical direction. The following parameters are used to define the non-dimensional quantities in the above governing equations and later physical explanations. u, v are the velocity variables in the x-, y-direction, respectively. p is the pressure, Re is the Reynolds number, $Pe=\frac{DU}{k}=\mathrm{Re}\,\mathrm{Pr}$ is the Peclet number, Pr denotes the Prandtl number.

3. Finite element scheme

In order to discretize the time derivative of the convection equation, consider the following equation,

$$\frac{\partial \phi}{\partial t}+u\frac{\partial \phi}{\partial x}+v\frac{\partial \phi}{\partial y}=0 \tag{5}$$