Available online at www.sciencedirect.com

ScienceDirect

Computers and Structures 86 (2008) 627–641

Computers & Structures

www.elsevier.com/locate/compstruc

Complete flow field computation around an ACV (air-cushion vehicle) using 3D VOF with Lagrangian propagation in computational domain

A.H. Nikseresht [a,*,1], M.M. Alishahi [b], H. Emdad [b]

[a] Department of Mechanical Engineering, Shiraz University of Technology, Shiraz, Iran
[b] Department of Mechanical Engineering, Shiraz University, Shiraz, Iran

Received 3 December 2006; accepted 15 August 2007
Available online 23 October 2007

Abstract

In this study an algorithm and a 3D solver is developed to solve the flow field around air-cushion vehicles (ACV) in vicinity of free surface. A single set of dimensionless equations is derived to handle both liquid and air phases in viscous 3D incompressible free surface flows in general curvilinear coordinates. The momentum equations are solved using the SIMPLE method in a staggered grid. The Lagrangian approach in the computational domain is also applied in the context of the VOF method to resolve the free surface. To demonstrate the robustness and versatility of the code, an application of this method on the impact problem of a circular cylinder is presented that is compared to experimental, theoretical and other computational results. In a 3D problem, the gravity-fill test is solved in both Cartesian and randomly generated curvilinear grid system. Comparison of results with the available experimental data and other numerical results is presented as well. At last, the code is applied to solve the two and three dimensional air and water flow field around an ACV and the effects of several parameters including the under skirt pressure distribution, initial air gap and effect of Fr. number on the wave elevation and wave making drag are investigated. It is shown that the presented method and the code are, robust and accurate enough to produce complicated 3D flow fields around ACV's.

Keywords: VOF; 3D; ACV; Impact; Fr. number; Wave-drag

1. Introduction

The generic title 'hovercraft' is used to cover the two principal types of marine vehicles discussed in the literature, namely air-cushion vehicle (ACV) and surface effect ship (SES). Each type obtains a vertical supporting force by generating an over-pressure (above atmospheric pressure) in a region between the keel and water surface, which literally lifts the main hull clear of water. The present day, ACV has, in principle, no part of the craft in contact with the land or sea surface and is, therefore, completely amphibious. Air is supplied from a lift fan, which provides airflow round the periphery of the hull followed by ejection into the cushion space. One of the most important objectives in modeling the flow field around the ACV is to obtain the total vehicle drag. The hydrodynamic aspects of an air-cushion vehicle (ACV) has been studied by assuming its action to be equivalent to that of a pressure distribution acting on the free surface of the water. This idealization prohibits any physical contact of the lower edge of the craft with the water. In addition, it has been assumed that the flow of escaping air under the periphery is inviscid, and therefore, produces no spray. Havelock, in some of his early papers [1,2] was the first to treat the theoretical problem of the wave resistance of a pressure dis-

* Corresponding author. Tel./fax: +98 711 7264102.
E-mail addresses: nikser@sutech.ac.ir (A.H. Nikseresht), alisha@shirazu.ac.ir (M.M. Alishahi), hemdad@shirazu.ac.ir (H. Emdad).
[1] Assistant Professor.

0045-7949/$ - see front matter © 2007 Elsevier Ltd. All rights reserved.
doi:10.1016/j.compstruc.2007.08.006

tribution, but most of the distributions that he chose to analyze were very smooth and were not typical of the ACV. Barratt [3] also computed the wave resistance of rectangular and elliptical pressure distributions, but for the case of unrestricted water. Doctors [4] has found it necessary to "smooth off" pressure distribution in an artificial and empirical manner near the edges, in order to obtain finite wave resistance. Tuck [5] considered the flow of air above stationary water, and flow of water below air at atmospheric pressure.

In these studies the details of the flow field around the body of ACV are neglected. In recent years modeling of the free surface flows is extended to viscous flows. However, most of the investigations have been directed toward fluid flow solution around ships [6,7]. Therefore, the main contribution of the present work is to simulate the complete viscous free surface flow including 3D air flow around the ACV.

The available numerical methods for such problems can be classified into moving and fixed grid approaches. The fixed grid approach seems to be a more viable method whenever a general motion of free surface flow is considered. Among the existing fixed grid approaches, Harlow and Welch [8] proposed the well-known marker and cell method (MAC). The MAC method defines the fluid region rather than the free surface, and thus requires large computer storage and additional computational time to move all fluid markers to new locations especially when a three-dimensional problem is encountered. Furthermore, a finite volume far from the free surface might be unrealistically overfilled or partially filled with markers due to numerical errors. In 1981, Hirt and Nichols [9] introduced the volume of fluid method (VOF) for incompressible flow with a moving free surface.

In the VOF method, the interface describes implicitly, the data structure that represents the interface in terms of C, which is the fraction of each cell that is filled with a reference phase, say phase 1. The scalar field C is often referred to as the color function. The magnitude of C in the cells cut by the free surface is between 0 and 1 ($0 < C < 1$) and away from it, is either zero or one. Typically, one can reconstruct the interface by the straightforward simple line interface calculation (SLIC) method [10] or by various "piecewise linear interface calculation" (PLIC) methods [11–13]. The latter give much better results than the former. Some recent works that implement VOF methods in curvilinear coordinates are presented by Kothe et al., [14]. Their methods that are carried out in physical domain have some difficulties that cannot be easily applied in conjunction with operator split techniques.

Fluid–solid interaction (FSI) has been the foci of several studies in recent years. To give some examples, [15,16] review the state of the art of fluid–solid interaction as applied in a standard software package. Several examples including (FSI) are contained in the above mentioned references and also in other references not mentioned here. Free stream flows in an extensive domain such as the present problem are not covered in [15,16]. The large deformation of an ACV skirt and its interaction with the underneath jet flow presents an interesting FSI problem. Using commercial softwares, the computation of such FSI problems would be a burden due to the large flow field dimensions and extensive computer resources required. Using the especially prepared finite volume solver together with a large deformation finite element solver for the skirt in an interactive environment provides a tailor made software with unique capabilities. This study focuses on the first part of such a package program.

In the present study, the finite volume method in physical domain is used to solve the Navier–Stokes equations. However, the free surface equation is transferred to the computational domain and is solved in that domain. The two steps of propagation and reconstruction of the free surface are carried out in this domain. In the present work the PLIC method for the interface reconstruction is used. The main advantages of the proposed technique, i.e., PLIC–VOF method in computational domain over the earlier method of PLIC in physical domain are explained in the previous articles by the present authors [17–19]. Therefore, the other contribution of the present study is the implementation of the above mentioned technique in a complicated flow field such as 3D flow around an ACV.

Incompressible Navier–Stokes equations are discretized using finite volume method based on pressure correction algorithm (SIMPLE) [20]. Hence, the appropriate numerical algorithm to solve the Navier–Stokes equations for a two-phase incompressible flow in curvilinear coordinates with high-density difference is introduced. Performance of the proposed numerical procedure is examined through solution of 2D problems such as a well-documented dam-breaking example in Cartesian and curvilinear coordinates and an impact problem of a circular cylinder [18]. In a 3D problem the gravity-fill test in both Cartesian and a randomly generated curvilinear grid system is solved. Comparison of results with the available experimental and results from other numerical methods is presented as well. At the end the code is used to solve two and three dimensional air and water flow field around an ACV and the effects of changing several parameters, including under the keel pressure distribution, initial gap, and Fr. number are investigated.

2. Problem formulations

2.1. Governing equations

Let $\vec{U}$ be the velocity vector field, ρ the density, P the pressure, μ the viscosity, n the unit normal to the interface, the Navier–Stokes equation then reads:

$$\partial_t(\rho\vec{U}) + \nabla\cdot(\rho\vec{U}\otimes\vec{U}) = -\nabla p + \nabla\cdot(2\mu\vec{D}) + \rho\vec{g} \qquad (1)$$

where D is the rate-of-strain tensor with components