Computers and Fluids 125 (2016) 116–129

Contents lists available at ScienceDirect

Computers and Fluids

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

A multi-region coupling scheme for compressible and incompressible flow solvers for two-phase flow in a numerical wave tank

P.J. Martínez Ferrer*, D.M. Causon, L. Qian, C.G. Mingham, Z.H. Ma

Centre for Mathematical Modelling and Flow Analysis, Manchester Metropolitan University, Chester Street, Manchester M1 5GD, United Kingdom

ARTICLE INFO

Article history:
Received 9 February 2015
Revised 2 September 2015
Accepted 18 November 2015
Available online 2 December 2015

Keywords:
Hydrodynamics
Coupling
Compressibility
Aeration
Wave impact

ABSTRACT

We present a multi-region coupling procedure based on the finite-volume method and apply it to two-phase hydrodynamic free surface flow problems. The method combines the features of one incompressible and one compressible two-phase flow solvers to obtain a coupled system which is generally superior to either solver alone. The coupling strategy is based on a partitioned approach in which different solvers, pre-defined in different regions of the computational domain, exchange information through interfaces, i.e. areas separating these regions. The interfaces act as boundary conditions passing the information from one region to the other mimicking the finite-volume cell-to-face interpolation procedures. This results in high performance computing coupled simulations whose functionality can be further extended in order to build a generic numerical wave tank accounting for incompressible flow regions as well as compressibility and aeration effects. We select a series of preliminary benchmarks to verify this coupling procedure which includes the simulation of a hydrodynamic dam break, the propagation and reflection of regular waves, the convection of an inviscid vortex, pseudocavitation, a water column free drop in a closed tank and a plunging wave impact at a vertical wall. The obtained results agree well with exact solutions, laboratory experiments and other numerical data.

1. Introduction

Coupled simulations, in which different numerical solvers are working together, are in growing demand and present a challenging task. In recent years, an increasing number of coupled simulations have been employed in the scientific community and industry in very different disciplines such as Fluid-Structure Interaction or FSI [1–4], multi-physics [5,6] and multi-solver [7–13] problems and, in particular, incompressible–compressible coupling [14,15], to mention but a few. The range and complexity of problems are expanding while they vary widely in their nature, both in terms of the included physics and manner of coupling. In general terms, a coupling strategy consists of integrating together individual solvers with very specific features in order to improve the accuracy, reduce computational costs and extend the functionality of the global simulation. Due to the large diversity of coupling solutions, there is no general implementation of these strategies. Moreover, with the increasing demand on High Performing Computing (HPC) and heterogeneous computing, e.g. GPGPU and hybrid parallel programming, some strategies remain inefficient as they cannot easily be implemented in parallel via domain decomposition methods [3,8,13]. In some cases, like the one presented in Ref. [10], the interface between the solvers is in charge of communicating the relevant information using the message passing interface (MPI) protocol allowing the running of coupled simulations on massively parallel supercomputers.

The approach presented in this work aims to overcome these issues by introducing a procedure based on the finite-volume method. We are particularly interested in the coupling of pre-defined finite-volume meshes, each one containing its specific solver, within a unique global simulation by specifying an appropriate set of boundary conditions at the interface connecting the meshes, also referred to as regions. For this purpose, we rely on a "partitioned approach", e.g. see Refs. [8,12], in which each region is simulated separately and the coupling is performed at the interface. The main advantages of the partitioned approach compared to other approaches, such as the "monolithic approach" utilized in Ref. [13], are its easy implementation and wide range of coupling options: the individual solvers do not need to be extensively modified allowing, for instance, coupling of finite-element and finite-volume based solvers in a relatively easy manner. For this reason, the partitioned approach has been popularly adopted in FSI simulations [3] and hybrid model coupling [9].

Our ultimate goal is to construct a Virtual Wave Structure Interaction (WSI) Simulation Environment such as the one schematized in Fig. 1. A Numerical Wave Tank (NWT) is used to evaluate the impact of waves on fixed or floating objects, offshore structures as well as to assess the performance of wave energy converters. Traditionally, NWTs

* Corresponding author. Tel.: +44 1612471519.
E-mail address: p.martinez-ferrer@mmu.ac.uk (P.J. Martínez Ferrer).

http://dx.doi.org/10.1016/j.compfluid.2015.11.005
0045-7930/

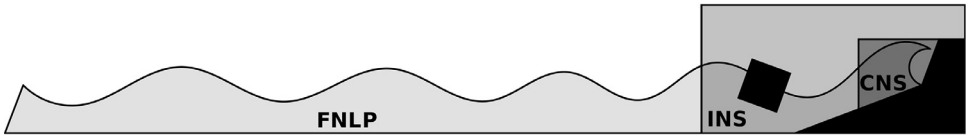

Fig. 1. Virtual Wave Structure Interaction (WSI) Simulation Environment.

have been constructed using potential flow models [16], incompressible Navier-Stokes solvers [17,18] or a combination of both [8,13] for engineering design and analysis. However, in violent wave impacts and slamming events the fluid compressibility needs to be taken into account for the correct prediction of impact loadings [19]. Additionally, aeration effects may become important in hydrodynamics problems involving enclosed air bubbles and air pockets trapped in water bodies [20]. Therefore, the incompressible assumption of the fluids, both air and water, in which the influence of air on waves is ignored because of its small density compared to water, must be revisited. In fact, especial attention has begun to be paid to the compressibility of the water–air mixture in recent years [21–25]. Furthermore, experiments of wave impacts on structures have demonstrated that during the temporal transition of a plunging wave, the air pocket trapped in the water body expands and compresses yielding a strong peak pressure and subsequent pressure oscillations above and below the ambient atmospheric pressure which can potentially damage offshore structures [20,26,27]. Therefore, compressibility effects in the water–air mixture and air pockets as well as cavitation effects, i.e. change of phase, need to be handled properly. Thus, in order to represent the most accurate physics where most necessary within a NWT, specialized numerical solvers must be coupled together within the same simulation framework, e.g. using the scheme given in Fig. 1: (i) a Fully Non-Linear Potential (FNLP) solver generates and propagates accurately the waves from the wave generator source, (ii) an interface-capturing Incompressible Navier–Stokes (INS) solver is used when aeration and compressibility of the water–air mixture are negligible and (iii) an interface-capturing Compressible Navier Stokes (CNS) may be envisaged to evaluate accurately the expected violent wave loads against structures. Fixed structures and floating bodies may be deformable and thus a Computational Structural Dynamics (CSD) code may also be necessary for wave and structure interactions.

In this work we introduce a new coupling procedure based on the finite-volume method and concentrate our efforts on the coupling of the INS and CNS solvers which we validate through a series of numerical benchmarks. Section 2 describes the incompressible and compressible solvers, the coupling procedure and the solution procedure algorithm for the coupled simulations. Results and discussions are provided in Section 3 and Section 4 is dedicated to conclusions and future work.

2. Numerical procedures

The numerical methods used in this work rely on a cell-centered, co-located finite-volume method already implemented in the open-source CFD software OpenFOAM [28]. The reason for using this CFD library is because it is a robust and advanced tool widely employed in research and industry. Moreover, the advantage of being open-source allows to read, improve and modify the available code for free. Nevertheless, we stress the fact that the coupling strategy and numerical methods described below can be applied to any finite-volume solver in different engineering applications such aerodynamics, hydrodynamics, etc. We modify two native, incompressible and compressible, two-phase pressure-based solvers available in OpenFOAM in order to get them working together exchanging information at their interfaces through a new set of appropriate boundary conditions. A description of the solvers, the coupling strategy as well as the solution algorithm is detailed below.

2.1. The incompressible solver "interFoam"

The native OpenFOAM based solver "interFoam" is an incompressible two-phase pressure-based solver [29] which has successfully been applied in a wide variety of naval and coastal engineering applications, see for instance [30,31]. It solves the three-dimensional equations for two incompressible phases, i.e. air and water, using the volume of fluid (VOF) method with especial emphasis on maintaining a sharp free surface (interface-capturing solver).

The first equation to be solved is the mass balance. For an incompressible two-phase flow ($\nabla \cdot \boldsymbol{U} = 0$), only the mass balance equation for the water volume fraction $\alpha \in [0, 1]$ is considered

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot \boldsymbol{U}\alpha + \nabla \cdot \boldsymbol{U}_c \alpha(1-\alpha) = 0, \tag{1}$$

where $\boldsymbol{U}$ is the mixture velocity vector and $\boldsymbol{U}_c = \min[\boldsymbol{U}, \max(\boldsymbol{U})]$. The third term in Eq. (1) is an artificial compression term that helps to maintain the interface sharp and bounded between zero and unity using the MULES procedure [29,32].

The single momentum equation for the homogeneous mixture is given by

$$\frac{\partial \rho \boldsymbol{U}}{\partial t} + \nabla \cdot (\rho \boldsymbol{U}\boldsymbol{U}) - \nabla \cdot (\mu_{\text{eff}} \nabla \boldsymbol{U}) = \sigma \kappa \nabla \alpha - \boldsymbol{g} \cdot \boldsymbol{x} \nabla \rho - \nabla p_d, \tag{2}$$

where $\rho = \alpha \rho_w + (1-\alpha)\rho_a$ is the density of the mixture; note that the sum over the volume fractions of water and air is equal to unity and $\rho_w = 1000\,\text{kg/m}^3$ and $\rho_a = 1.1586\,\text{kg/m}^3$ are constant parameters. The surface tension coefficient is represented with the variable σ and the curvature of the interface is calculated as $\kappa = \nabla \cdot (\nabla\alpha/|\nabla\alpha|)$. Furthermore, the mixture viscosity is given by $4(\alpha\mu_w + (1-\alpha)\mu_a)/3$, $p_d = p - \rho \boldsymbol{g} \cdot \boldsymbol{x}$ is the piezometric pressure and $\boldsymbol{g}$ and $\boldsymbol{x}$ are the gravity and the position vectors, respectively. It can be noticed from Eq. (2) that heat and mass transfer are neglected. Additionally, a pressure correction equation is derived from the momentum and continuity equations to solve for the dynamic pressure and correct the velocity adopting a segregated approach.

2.2. The compressible solver "compressibleInterFoam"

This OpenFOAM native solver is an extension of the previous interface-capturing solver. The water phase is treated as a compressible perfect fluid, $\rho_w = \rho_{w,0} + p/R_w T$ with $\rho_{w,0} = 1000\,\text{kg/m}^3$ and $R_w = 3000\,\text{J/kg K}$, while the air phase is a compressible perfect gas, $\rho_a = p/R_a T$ with $R_a = 287\,\text{J/kg K}$. The water volume fraction transport equation presents an additional source term due to the compressibility of this phase:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot \boldsymbol{U}\alpha + \nabla \cdot \boldsymbol{U}_c \alpha(1-\alpha) = -\frac{\alpha}{\rho_w}\frac{\mathrm{D}\rho_w}{\mathrm{D}t}, \tag{3}$$

which is implicitly calculated from the pressure correction equation at the previous time step. The density of the compressible mixture is evaluated according to

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \boldsymbol{U}) = 0. \tag{4}$$

The same single momentum equation for the homogeneous mixture (2) is solved and an additional transport equation for the