Ocean Engineering xxx (xxxx) xxx–xxx

Contents lists available at ScienceDirect

Ocean Engineering

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

Enhanced coupling of solid body motion and fluid flow in finite volume framework

Inno Gatin[a,*], Vuko Vukčević[a], Hrvoje Jasak[a,b], Henrik Rusche[b]

[a] University of Zagreb, Faculty of Mechanical Engineering and Naval Architecture, Ivana Lučića 5, Zagreb, Croatia
[b] Wikki Ltd, 459 Southbank House, SE1 7SJ London, United Kingdom

A R T I C L E I N F O

Keywords:
6–DOF rigid body motion
Motion–fluid coupling
Seakeeping
Naval hydrodynamics
Foam–extend

A B S T R A C T

An enhanced coupling strategy for the resolution of 6–degrees–of–freedom rigid body motion and unsteady incompressible fluid flow in Finite Volume collocated arrangement using PIMPLE algorithm and rigid mesh motion is presented in this paper. The improved coupling is achieved by calculating the 6–degrees–of–freedom motion equations after each pressure correction step in the pressure–velocity PISO (Pressure Implicit With Splitting of Operators) algorithm. Solving the 6–degrees–of–freedom equations after each solution of the pressure equation of the PISO loop accelerates the convergence, leading to smaller number of nonlinear pressure–velocity iterations needed per time–step. The novel approach is verified and validated on a heaving decay case, while the achieved acceleration in terms of the number of PISO loops is demonstrated on seakeeping simulations of a container ship.

1. Introduction

Numerical simulations using Computational Fluid Dynamics (CFD) are frequently used in computational naval hydrodynamics for assessing wave induced loads and motions (Larsson et al., 2013, 2015a, 2015b). There are numerous reasons why wave induced loads and motions of floating objects are important in marine engineering. Fuel consumption of ships sailing in waves is one of them, due to the increase of oil price during the last couple of decades, as well as the increasingly rigorous regulations regarding harmful gas emission. Seakeeping characteristics of ships are important for safety and comfort of crew and passengers, as well as for assessing acceleration loads (e.g. heavy deck equipment, superstructures etc.).

CFD is proving to be a useful tool in predicting behaviour of ships in waves. Numerous publications (e.g. Orihara and Miyata, 2003; Carrica et al., 2008, 2011, 2012; Bhushan et al., 2009; Kim, 2011; Castiglione et al., 2011; Wu et al., 2011; Guo et al., 2012; Miyata et al., 2014; Mousaviraad et al., 2015; Sadat-Hosseini et al., 2013; Simonsen et al., 2013; Tezdogan et al., 2015) tend to depict the accuracy and potential of CFD for solving such problems, using different ways to couple 6–degrees–of–freedom (6–DOF) motion and fluid flow. The coupling of body motion and fluid flow is commonly performed on the level of the nonlinear pressure–velocity loop (SIMPLE or PIMPLE), i.e. after the flow solution rigid body motion equations are solved and the computational grid is moved accordingly. The procedure is then repeated within each time–step until convergence. This is the conventional strongly coupled approach, hereinafter referred to as conventional approach. The PIMPLE algorithm is comprised of multiple PISO pressure–velocity loops, where pressure is updated multiple times per one momentum equation–update (Issa, 1986).

The above mentioned, conventional approach has been verified in numerous publications. Orihara and Miyata (2003) use a predictor–corrector algorithm for the in–house code WISDAM–X, where they recalculate the entire flow field after every body motion correction. Castiglione et al. (2011) imply that the in–house code CFDShip-Iowa uses a similar approach, where the complete fluid flow solution is obtained in each body motion–fluid flow iteration. Wu et al. (2011) describe the execution sequence of the CFD code used in their study where a similar procedure is employed. To achieve convergence of the coupling, multiple body motion–fluid flow iterations are needed. Simonsen et al. (2013) and Vukčević and Jasak (2015) reported that a minimum of five pressure–velocity (PISO) loops were needed per time–step to ensure convergence. For the fluid flow itself to converge, smaller number of PISO loops is sufficient, typically two for wave related problems. Hence, the body motion–fluid flow coupling presents a considerable overhead in terms of CPU time.

A modified approach for coupling the rigid body motion equations and fluid flow is described, verified and validated in this paper. Pressure field and body motion are tightly coupled at the body boundary in large scale naval hydrodynamics problems. Pressure

* Corresponding author.
E-mail addresses: inno.gatin@fsb.hr (I. Gatin), vuko.vukcevic@fsb.hr (V. Vukčević), h.jasak@wikki.co.uk, hrvoje.jasak@fsb.hr (H. Jasak), h.rusche@wikki.co.uk (H. Rusche).

http://dx.doi.org/10.1016/j.oceaneng.2017.08.009
Received 6 December 2016; Received in revised form 13 June 2017; Accepted 7 August 2017
0029-8018/

influences the body motion through the force acting on the body, while the moving body influences the flow through the change of body boundary velocity and relative grid motion fluxes (see Demirdžić and Perić, 1988). Rigid grid motion is used, i.e. the grid is not deformed when the body is moving. Enhancing the coupling between the pressure equation and the body motion equations has been proposed before for resolving the coupling of fluid flow and elastic bodies (Fernández et al., 2005), however no similar approach is encountered for a special case of rigid bodies that are modelled as boundaries of the fluid domain, where no volume discretisation of the body is present.

In this work the convergence of the body motion–fluid flow coupling is accelerated by further resolving the coupling via updated 6–DOF solutions after each pressure correction equation within the PISO loop in addition to the standard motion update after each PISO loop. The grid position is not updated between every pressure correction in order to save CPU time. This is allowed since relative flux caused by the grid motion does not influence the pressure equation due to its elliptic nature for incompressible flows. Furthermore, the influence of the new grid position is considered negligible since the motions are generally small within a time–step, even for large overall motions (e.g. manoeuvres). We stress that the grid motion and relative fluxes are updated after each PISO loop in a given time–step, correctly accounting for the complete 6–DOF–fluid flow coupling. Tighter coupling leads to a smaller number of PISO iterations needed to ensure body motion–fluid flow coupling convergence, which in turn reduces the overall CPU time.

The benefit of the presented approach over the conventional approach is the tighter coupling of the pressure equation and the 6–DOF equations which dictate the motion of the body, which in turn represents the boundary of the fluid domain. In the conventional approach, the 6–DOF equations are solved once per PISO loop, i.e once per pressure–velocity coupling. In the proposed approach, 6–DOF equations are solved a significantly larger number of times: in addition to the standard update in every pressure–velocity coupling loop, the 6–DOF equations are additionally solved every time the pressure equation is solved.

This paper is organised as follows. First, the numerical model of the enhanced approach for fluid flow–6–DOF coupling is described, comprising the governing equations, brief description of the numerical procedure and a detailed procedure of the novel algorithm. Second, a test case of a heaving cylinder is presented to verify and validate the novel approach by comparing the results with experimental and analytical results. Next, container ship seakeeping test cases are presented to demonstrate the improvement of convergence of rigid body motion–fluid flow coupling achieved with the new approach, accompanied by a discussion of the results. Finally a brief conclusion is given.

2. Numerical method

The enhanced 6–DOF–fluid flow coupling scheme is implemented in foam–extend (Jasak, 2009), a community driven fork of OpenFOAM open source software, which uses second–order accurate finite volume spatial discretisation with arbitrary polyhedral grid support (Jasak and Gosman, 2001). In this section a brief overview of the discretised governing equations for incompressible two–phase flow is given. The numerical procedure based on the PISO algorithm including the solution of 6–DOF rigid body motion equations is shown. Finally, the novel approach for coupling 6–DOF body motion equations with the pressure equation is presented.

2.1. Fluid flow governing equations

In free surface hydrodynamic problems, the incompressible two–phase flow is governed by the momentum equation, continuity equation and the free surface transport equation. Two phases are modelled with a single set of governing equations, where the discontinuity in pressure gradient and density at the interface is resolved using the Ghost Fluid Method (GFM) (Vukčević, 2016; Vukčević et al., 2017). The GFM imposes pressure jump conditions at the free surface ensuring a sharp transition of fluid properties. For incompressible fluids the conservation of mass is governed by:

$$\nabla \cdot \mathbf{u} = 0, \tag{1}$$

where $\mathbf{u}$ represents a continuous velocity field in the global coordinate system. For a moving computational grid the momentum equation reads:

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot ((\mathbf{u} - \mathbf{u}_M)\mathbf{u}) - \nabla \cdot (\nu_e \nabla \mathbf{u}) = -\frac{1}{\rho}\nabla p_d, \tag{2}$$

where $\mathbf{u}_M$ is the relative grid motion velocity which stems from the Space Conservation Law (Demirdžić and Perić, 1988); ν_e is the effective kinematic viscosity comprising appropriate phase viscosity and turbulent eddy viscosity; ρ is the density field, and p_d stands for dynamic pressure: $p_d = p - \rho\mathbf{g} \cdot \mathbf{x}$. Note that due to the GFM, volumetric fluxes are used for convection instead of mass fluxes (see Vukčević et al., 2017 for details). Algebraic Volume of Fluid (VOF) (Rusche, 2002) method is used for interface capturing with additional convective term for interface compression:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\mathbf{u}\alpha) + \nabla \cdot (\mathbf{u}_r \alpha(1 - \alpha)) = 0, \tag{3}$$

where α is the volume fraction, and $\mathbf{u}_r$ stands for artificial compressive velocity field which is oriented in the normal direction towards the free surface (Weller, 2008). The third term is active only near the free surface due to the nonlinear term $\alpha(1 - \alpha)$. The details on the evaluation of $\mathbf{u}_r$ can be found in Rusche (2002).

Detailed discretisation of temporal derivative, convection and diffusion in (2) in integral form can be found in Jasak (1996), while already discretised equations are used in the text below. The semi–discretised momentum equation for each cell reads:

$$a_P \mathbf{u}_P + \sum_f a_N \mathbf{u}_N = \mathbf{b} - \frac{1}{\rho}\nabla p_d, \tag{4}$$

where a_P stands for the diagonal coefficient, a_N for the off–diagonal coefficients, and subscripts P and N stand for values in the parent cell centre and neighbouring cell centres, respectively. Parent cell is the cell for which the equation is being solved for, and the neighbouring cells are all the cells which share a face with the parent cell (Jasak and Gosman, 2001). $\sum_f$ is the sum over all neighbouring faces f, and $\mathbf{b}$ stands for the source term. Following notation proposed by Jasak (1996), (4) can be written as:

$$\mathbf{u}_P = \frac{H(\mathbf{u}_N)}{a_P} - \frac{1}{a_P}\frac{\nabla p_d}{\rho}, \tag{5}$$

where $H(\mathbf{u}_N)$ presents an explicit operator:

$$H(\mathbf{u}_N) = -\sum_F a_N \mathbf{u}_N + \mathbf{b}. \tag{6}$$

The pressure equation is derived by interpolating (5) on cell faces and substituting into the discretised form of (1), yielding:

$$\sum_f \mathbf{s}_f \cdot \left(\frac{1}{a_P}\right)_f \left(\frac{1}{\rho}\nabla p_d\right)_f = \sum_f \mathbf{s}_f \cdot \frac{(\mathbf{H}(\mathbf{u}_N))_f}{(a_P)_f}, \tag{7}$$

where $\mathbf{s}_f$ stands for surface normal vector, and subscripts f denote values at face centres. For a detailed derivation of the pressure equation with the GFM, the reader is referred to Vukčević et al. (2017).

2.2. Numerical procedure

The solution of above equations is achieved in a segregated manner in a PIMPLE loop, a combination of SIMPLE and PISO algorithms,