ACOMEN 2017

September 18-22, Ghent, Belgium

GHENT
UNIVERSITY

Book of Abstracts

Editors:
Marián Slodička, Ghent University
Karel van Bockstal, Ghent University

Organising committee:
C. Geuzaine (Univ. de Liège, Belgium)
S. Pop (UHasselt, Belgium)
K. Van Bockstal (Ghent Univ., Belgium)

Scientific committee:
J. Kačur (Comenius Univ., Bratislava, Slovakia)
G. S. Dulikravich (Florida Inte. Univ., USA)
P. Šolín (University of Nevada, Reno, USA)
S. Majid Hassanizadeh (Utrecht University, The Netherlands)
E. Zuazua (Universidad Autónoma de Madrid, Spain)
C. Geuzaine (Univ. de Liège, Belgium)
S. Pop (UHasselt, Belgium)

Sponsors and partners:

ISBN: xxxxxx

Contents

Book of abstracts of the 7th International Conference on Advanced Computational Methods in Engineering, ACOMEN 2017 18–22 September 2017.

Accelerated numerical simulations of a heaving floating body by coupling a motion solver with a two-phase fluid solver

Brecht Devolder[1,2,*], Peter Troch[1] and Pieter Rauwoens[2]

[1] *Department of Civil Engineering, Ghent University*

[2] *Department of Civil Engineering, Technology Cluster Construction, KU Leuven*

e-mails: Brecht.Devolder@UGent.be, Peter.Troch@UGent.be, pieter.rauwoens@kuleuven.be

Abstract

Numerical simulations of floating bodies, heaving under wave loading, are performed by coupling a fluid solver and a motion solver. During each time step in the transient simulation, a converged solution is needed between the wave-induced fluid field around and the motion of the body. For some geometries in particular, a very slow converging or unstable solution is found. The mechanism for this non-physical instability is identified and an accelerated coupling scheme is derived for speeding up the simulations.

Key words: CFD, two-phase flow, rigid body motion, accelerated coupling scheme
MSC 2010: 65L20, 70E15, 76T10

1 Introduction

This paper presents a study on the coupling between a fluid solver and a motion solver by interchanging the total force F acting on a floating body. The two-phase fluid solver with dynamic mesh handling, interDyMFoam, is a part of the Computational Fluid Dynamics (CFD) toolbox OpenFOAM. The incompressible Navier-Stokes (NS) equations are solved together with a conservation equation for the Volume of Fluid (VoF). In this paper, that fluid solver is coupled to a rigid body motion solver, restricted to the heave motion only.

We were able to identify the mechanism for a numerical instability between the fluid and motion solver and to derive an accelerated coupling scheme, which are explained below.

2 Governing equations

The acceleration of the body a is derived from Newton's second law: $F = ma$ and subsequently integrated to the velocity v and position z respectively. In general, the fluid solver calculates the total force F by a discrete sum of the pressure forces p and the viscous forces τ over the faces of the body completed with body's total weight. In order to identify the source of the instability, the total force acting on the body is not calculated by solving the NS equations. Instead, a 1D simplified mass-spring-damper system is used in which the damping contribution is neglected ($b = 0$). Its equation of motion is explicitly formulated in terms of a by:

$$F^{n+1} = ma^{n+1} = -m_a a^n - bv^n - k(z^n - z^{eq}) \tag{1}$$

$$a^{n+1} = -\frac{m_a}{m}a^n - \frac{k}{m}(z^n - z^{eq}) \tag{2}$$

in which m_a is the added mass, b the damping coefficient, v the body's velocity, k the spring constant representing the hydrostatic restoring force and $z^n - z^{eq}$ the distance between the Centre of Mass (CoM) at the previous time step n and the CoM in equilibrium.

A linear stability analysis proves that (2) is only stable if $m_a < m$. Otherwise, an implicit formulation is needed for a (3a) using a relaxation method with relaxation factor α (3b).

$$a_{i+1}^{n+1} = -\frac{m_a}{m}a_i^{n+1} - \frac{k}{m}(z_i^{n+1} - z^{eq}) \tag{3a}$$

$$a_{i+1}^{n+1} = \alpha a_{i+1}^{n+1} + (1-\alpha)a_i^{n+1} \tag{3b}$$

in which $i+1$ and i are the current and previous iteration during the same time step $n+1$.

3 Accelerated coupling scheme

The coupling between motion and fluid solver is accelerated by using an optimal value for α (4a). Consequently, only one sub-iteration is needed for (3b) if m_a is known. However, in general m_a is unknown and varies from time to time (e.g a floating body in an irregular sea-state). Therefore Newton's second law $F = ma$ is linearised and m_a is estimated by (4b).

$$\alpha_{optimal} = \frac{1}{1 + m_a/m} \tag{4a}$$

$$m_a = -\frac{F_i^{n+1} - F_{i=1}^{n+1}}{a_i^{n+1} - a_{i=0}^{n+1}} \tag{4b}$$

As a result, the new acceleration is obtained by substituting (4b) in (4a) and apply (3b). As an example, the accelerated scheme is applied for a free decay test of a 2D floating block for which $m_a = 3m$ and compared to (3b) using α equal to 0.05 and 0.45. It is clearly shown in Fig. 1, that (4a) accelerates the convergence significantly between the fluid and motion solver.

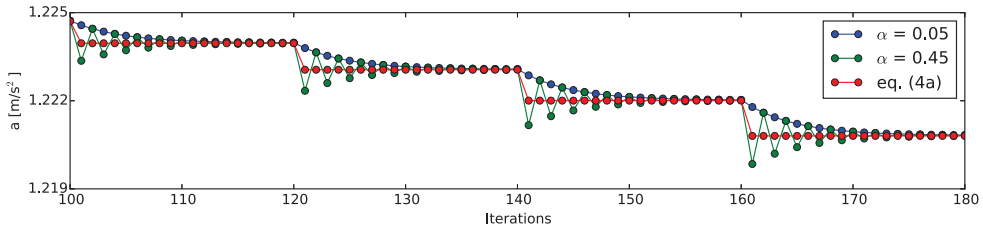

Figure 1: Acceleration as a function of the number of iterations for different values of α and eq. (4a). During each time step, 20 iterations are performed.

Acknowledgements

The first author is Ph.D. fellow of the Research Foundation-Flanders (FWO), Belgium (Ph.D. fellowship 1133815 N).