Journal of Fluids and Structures 81 (2018) 528–0

Contents lists available at ScienceDirect

Journal of Fluids and Structures

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

Added mass evaluation with a finite-volume solver for applications in fluid–structure interaction problems solved with co-simulation

C. Yvin [a,b], A. Leroyer [b,*], M. Visonneau [b], P. Queutey [b]

[a] SIREHNA/Naval Group, Technocampus Ocean, 5 rue de l'Halbrane, 44340 Bouguenais, France
[b] Laboratoire de recherche en Hydrodynamique, Energétique et Environnement Atmosphérique, École Centrale de Nantes, CNRS-UMR 6598, 44321 Nantes Cedex 3, France

ARTICLE INFO

Article history:
Received 10 January 2018
Received in revised form 16 March 2018
Accepted 4 May 2018

Keywords:
Fluid–structure interaction
Cooperative simulation
Added mass
Artificial added mass
Added mass evaluation
Code coupling
Partitioned approach

ABSTRACT

This work is devoted to the implementation and analysis of an efficient fluid–structure interaction coupling algorithm used in a cooperative simulation context for rigid bodies. This framework makes possible the study of complex fluid–structure configurations, through the coupling between CFD and dedicated multibody dynamics solvers.

A specific focus is here laid on the characterisation of the numerical properties of the proposed coupling algorithm in terms of convergence speed and stability. In order to stabilise the segregated algorithm while keeping non-intrusive in the structure solver, a relaxation operator based on an artificial added mass technique is used. To compute an efficient relaxation operator, an original resolution of the added mass effect is implemented into the fluid finite-volume solver. Good convergence properties are observed for rigid bodies with six degrees of freedom even in case of strong destabilising added mass effects.

1. Introduction

Co-simulation (Co-operative simulation) is a simulation methodology which uses several numerical solvers to work together in order to solve a global problem (Jürgens, 2009). It becomes possible and interesting because current numerical methods are mature enough to solve complex models in specific science fields (structural dynamics, fluid dynamics, thermodynamics, combustion, electromagnetism, etc.). Thus, the use of existing and assessed solvers looks an appealing and pragmatic way to handle multi-physics configurations.

For example, in naval architecture, Computational Fluid Dynamics (CFD) solvers are used more and more to check and/or improve ships performances in resistance, propulsion, manoeuvrability and seakeeping (Queutey et al., 2012). Because CFD solvers are dedicated and optimised to the resolution of the fluid problem, they are not developed to handle complex mechanical systems like driven appendages, flexible floating wind turbines, wave energy converter, moored ships or towed underwater systems for instance. When the mechanical systems are complex, Computational Structural Dynamics (CSD) solvers can be used, but like CFD solvers, they are not dedicated to solve the whole problem. Indeed, in CSD solvers, the fluid loading is often replaced by analytical models which are most of the time too simple to accurately address the complexity of the flow physics (Masarati, 1999). To fully and accurately solve the whole problem when both fields are complex and highly coupled, co-simulation is attractive due to the re-use of existing software including both advanced physical models and a very strong numerical adaptation for each elementary problem (Kassiotis, 2009).

Co-simulation is not only a method used by scientists and engineers to solve a general problem but an efficient work method too. Indeed, the use of different solvers allows different development teams to collaborate. From a practical point of view, the validation and maintenance of the solvers are facilitated (Hou et al., 2012) because the problem is clearly divided.

https://doi.org/10.1016/j.jfluidstructs.2018.05.008
0889-9746/

However, the communication between each solvers must be performed at specific moments to be efficient. Thus, particular attention should be paid to coupling strategies to be both accurate and robust.

In Fluid–Structure Interaction (FSI), the problem is commonly divided to three problems (the fluid, the structure and the interface) but this is not always the case. Indeed, an FSI problem can be seen as a unique global problem and the resolution can be performed with a monolithic approach (Hübner et al., 2004; Heil, 2004; Dettmer and Perić, 2007; Saksono et al., 2007; Papadakis, 2008; Wick, 2011). This approach is difficult to implement in a co-simulation context because major modifications in each solver have to be implemented. Moreover the resolution is not an easy task because there are a lot of different unknowns of different nature, which, being partially coupled, have to be solved at the same time (Cervera et al., 1996). For the classical FSI problem that we are interested in, the parametrisation of the problem should contain the pressure, the velocity, the turbulence as well as the free surface unknowns, the mesh and the structural kinematics and the structural internal forces.

The partitioned approach (or block-iterative method Cervera et al., 1996) is most often used because of its ease of implementation. It consists in solving the fluid and structure problems in a segregated way. This approach is used here because it can be easily applied in a co-simulation context. However, the convergence may be slow (Degroote et al., 2009) and the coupling algorithm stability is not guaranteed due to the added mass effect (Belanger et al., 1995; Wall et al., 2006; Söding, 2001) (cf. Section 5). In order to stabilise the coupling algorithm, the artificial added mass method is sometimes used (Leroyer, 2004). This technique, which consists in increasing artificially the inertia of the structure and adding an acceleration dependent force, must be modified in a co-simulation context to avoid being too intrusive (cf. Section 5). In this paper, it is shown that the use of a physical relaxation operator based on the knowledge of the added-mass leads to an optimal convergence of the coupling in addition to be fully integrated, since the number of non-linear iterations to solve the fluid flow with FSI is similar to the one while imposing the motion (cf. Section 6). At the same time, the present work proposes an original approach to compute the classical added-mass operator when the frequency tends to infinite which overrides the limitations of classical linear potential solvers.

In this work, two different solvers are used. The first one solves the fluid problem and the other solves the mechanical behaviour. They are ISIS-CFD and MBDyn. Because of the abilities of both solvers, complex mechanical systems which strongly interact with fluids in naval architecture can be accurately studied.

Sections 2 and 3 briefly describe the two solvers. Section 4 introduces the FSI problem and the coupling algorithm used. Section 5 describes how the added mass effect is taken into account in the coupling algorithm. Section 6 explicits the numerical evaluation of the added mass effect in the framework of a finite-volume approach. Finally, two different applications are proposed in Section 7.

2. ISIS-CFD solver

ISIS-CFD is available as a part of the FINE^{TM}/Marine computing suite which is dedicated to marine applications. This is an incompressible unsteady Reynolds-averaged Navier–Stokes (RANS) solver developed by ECN-CNRS (Queutey et al., 2012). This solver is based on a cell-centred unstructured finite-volume method. Pressure–velocity coupling is obtained through a Rhie & Chow SIMPLE-type method. Free-surface flow is addressed with an interface capturing method, by solving a convection equation for the volume fraction of water, which is discretised with specific compressive discretisation schemes (Queutey and Visonneau, 2007). An Arbitrary Lagrangian Eulerian (ALE) formulation is used to take into account modification of the fluid spatial domain (Leroyer et al., 2008). It is associated with robust and fast grid deformation techniques (Leroyer and Visonneau, 2005). The temporal discretisation scheme is the Backward Difference Formula of order 2 (BDF2) when dealing with unsteady configurations. For each time step, an inner loop (denoted by non-linear loop) associated to a Picard linearisation is used to solve the non-linearities of the system.

3. MBDyn solver

MBDyn (Multi-Body Dynamics), is an open-source solver under the GNU GPL license developed at the *Dipartimento di Ingegneria Aerospaziale* of the *Politecnico di Milano*. It is intended for the simultaneous solution of multi-discipline problems including non-linear dynamics, aero-servo-elasticity, smart piezo-structural components, electric and hydraulic components. It is aimed at the modelling of complex systems (Ghiringhelli et al., 1999).

To solve the kinematic laws of a multi-body mechanical system, the Redundant Coordinate Set (RCS) formulation is used. This means that every inertial body has six rigid body Degrees of Freedom (DOF) even if they are constrained by joints for instance. Additional holonomic or nonholonomic constraint equations are added which introduce algebraic unknowns that are analogous to the Lagrange multipliers and directly represent the reaction forces and couples (Masarati, 1999). All these equations are written in the form of a set of first order Algebraic Differential Equations (ADE). Thus every rigid body is represented by 12 equations (6 equations that represent the momenta and 6 others the Newton's law that links the rate of changes in momentum due to forces). In this work, the BDF2 scheme is used. The ability to take into account multidisciplinary complex systems and the simplicity of implementation are the main advantages of this formulation.