European Journal of Mechanics B/Fluids 49 (2015) 272–286

Contents lists available at ScienceDirect

European Journal of Mechanics B/Fluids

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

A matrix free, partitioned solution of fluid–structure interaction problems using finite volume and finite element methods

R. Suliman [a,b,*], O.F. Oxtoby [a], A.G. Malan [a,c], S. Kok [b,d]

[a] Aeronautic Systems, Council for Scientific and Industrial Research, Pretoria, 0002, South Africa

[b] Department of Mechanical and Aeronautical Engineering, University of Pretoria, Pretoria, 0002, South Africa

[c] Department of Mechanical Engineering, University of Cape Town, Cape Town, 7700, South Africa

[d] Advanced Mathematical Modelling, Modelling and Digital Science, Council for Scientific and Industrial Research, Pretoria, 0002, South Africa

ARTICLE INFO

Article history:
Received 29 October 2013
Received in revised form
11 September 2014
Accepted 13 October 2014
Available online 24 October 2014

Keywords:
Fluid–structure interaction
Partitioned solution
Arbitrary Lagrangian–Eulerian (ALE)
Finite volume methods
Finite element methods
Parallelisation

ABSTRACT

A fully-coupled partitioned finite volume–finite volume and hybrid finite volume–finite element fluid–structure interaction scheme is presented. The fluid domain is modelled as a viscous incompressible isothermal region governed by the Navier–Stokes equations and discretised using an edge-based hybrid-unstructured vertex-centred finite volume methodology. The structure, consisting of a homogeneous isotropic elastic solid undergoing large, non-linear deformations, is discretised using either an elemental/nodal-strain finite volume approach or isoparametric Q8 finite elements and is solved using a matrix-free dual-timestepping approach. Coupling is on the solver sub-iteration level leading to a tighter coupling than if the subdomains are converged separately. The solver is parallelised for distributed-memory systems using METIS for domain-decomposition and MPI for inter-domain communication. The developed technology is evaluated by application to benchmark problems for strongly-coupled fluid–structure interaction systems. It is demonstrated that the scheme results in full coupling between the fluid and solid domains, whilst furnishing accurate solutions.

1. Introduction

While the field of Computational Mechanics has traditionally been dominated by structural and fluid modelling in isolation, many problems of interest are in truth strongly dynamic systems where there is a close coupling between fluid and solid domains. Fluid–Structure Interaction (FSI) modelling is a branch of Computational Mechanics which aims at accurately calculating these effects in a quantitative manner. For example, in aeroelastic systems, one important phenomenon is non-linear flutter response which has spawned the field of Computational Aeroelastics [1,2]. Computational Biomechanics is another major area in which complex dynamic structural responses are intimately coupled with fluid flow in cardiac, arterial and respiratory systems [3–6]. Other examples of such problems include structural loads on ships [7], flow induced vibrations in nuclear power plants [8] and wind response of buildings [9]. Though recent years have seen much research going into the development of FSI modelling technology [10–14], the efficient and robust modelling of large-scale, strongly-coupled systems which involve complex geometries is still a work in progress. In this paper, we develop and evaluate a fully-coupled, matrix-free methodology as a contribution towards this challenge. This is incorporated into the *Elemental*[1] flow solver developed for multi-physics applications [15].

This work focuses on FSI systems where there are strong interactions between the fluid and structural domains and weakly-coupled methods are, therefore, not considered as they may diverge or result in inaccurate solutions [16–19]. As a result, recent research in the field has been devoted to developing strongly-coupled modelling technologies. Strongly-coupled methods can be sub-classified into separate or partitioned and single or monolithic solution methods. The advantage of a monolithic over a partitioned

* Correspondence to: Department of Applied Mathematics and Theoretical Physics, University of Cambridge, Wilberforce Road, Cambridge, CB3 0WA, United Kingdom.

E-mail address: R.Suliman@damtp.cam.ac.uk (R. Suliman).

http://dx.doi.org/10.1016/j.euromechflu.2014.10.002
0997-7546/

[1] *Elemental* referred to in this paper was a scientific toolbox founded by A.G. Malan and has been deprecated in its entirety. A new Elemental™ has since been developed at Univ. of Cape Town which is being commercialised by Elemental Numerics (Pty) Ltd.

approach is that all the equations are considered simultaneously and a single system of equations is solved, which ensures stability and convergence. However, this approach may suffer from ill conditioning and convergence is generally slow [16]. The advantage of a partitioned approach is that it allows the use of two independent solution techniques for the fluid and solid equations in isolation. The drawback of partitioned approaches is that they generally require a separate coupling algorithm or additional outer iterations between the fluid and solid to achieve strong-coupling, which places an additional computational cost on the scheme [16,20,21]. The most popular partitioned coupling algorithms use fixed-point iteration methods or interface Newton–Krylov methods [22,20]. Fixed-point methods generally make use of Gauss–Seidel iterations which are slow to converge and methods to accelerate convergence, including Aitken and steepest descent relaxation and coarse-grid preconditioning, have been used [22,19, 23,24,21]. The Newton–Raphson methods require the computation of Jacobians, which may be difficult to compute exactly and various methods have been developed that use approximate Jacobians [25–27].

For the purpose of this work the fluid and structural domains are to be solved in a strongly-coupled partitioned manner, where the transfer of information occurs at solver sub-iteration level leading to a tighter coupling than if the subdomains are converged separately and negating the need for a separate coupling algorithm. This leads to a fully-converged solution at each timestep where both dynamic and kinematic continuity – i.e. continuity of forces and velocities – are satisfied at the fluid/solid interface. Our approach allows independence with respect to spatial discretisation of the fluid and solid domains. Many recent FSI efforts have made use of a single discretisation scheme, either finite volume [28,17, 29,30] or finite element [16,18,31–34], to solve the entire domain, which simplifies the treatment at the interface of the fluid and solid domain. However, each method contains certain inherent advantages and should be used as such. In this work we have chosen to use the finite volume method for the fluid domain. For the solid domain, we use both the finite element method and an enhanced finite volume method [35,36]. The enhanced finite volume method is essentially a hybrid between the traditional node-based finite volume method, which suffers from locking with high aspect-ratio elements [35], and the element-based strain method [29], which suffers from odd–even decoupling. The fluid physics is described with the incompressible Navier–Stokes equation written for an Arbitrary Lagrangian–Eulerian (ALE) coordinate frame. A total Lagrangian formulation is employed in the case of the solid. This prevents discretisation errors from accumulating over time. Dynamic unstructured fluid mesh movement technology is developed using a simple interpolation method in the interests of computational efficiency and parallelisability. For the finite volume method, the non-linear, unified governing equations are spatially discretised via a compact unstructured, edge-based finite volume method whose spatial accuracy is notionally of second order. In the interests of both computational and programming efficiency, the chosen spatial discretisation algorithm should be naturally applicable to any part of a fluid or solid mesh. In the case of the fluid, this is achieved by employing an edge-based compact [37,38] discretisation methodology, which holds the additional advantage of being computationally considerably more efficient than element-based approaches [39] while being ideally applicable to massively parallel distributed memory machines. For the solid domain, we use both the enhanced finite volume method, which is a hybrid of the traditional node-based approach and the element-based strain method, as well as a higher-order Q8 finite element solver. The coupling of a higher-order finite element formulation for the structure and a linear finite volume formulation for the fluid leads to non-matching nodes at the fluid/solid interface and the transfer of information at these nodes will also be addressed in this paper. Dual-timestepping [40] is employed for the purpose of temporal discretisation.

As noted, the proposed partitioned modelling method allows for complete flexibility in terms of the solution strategy employed for the fluid and solid domains as these contain widely varying characteristic velocities—the fluid may be incompressible while the solid may range from compressible to almost incompressible. In the case of the fluid, the Artificial Compressibility Characteristic Based Split (CBS-AC) algorithm [41–43] is used. This scheme combines two historically opposing methodologies viz. *pressure based* (pressure projection—PP) proposed by Patankar [44] and the *density based* (artificial-compressibility—AC) method introduced by Chorin [45]. The scheme allows matrix-free solution of compressible as well as incompressible flows, which is of key value to large scale distributed memory computing. In this work the CBS-AC algorithm has been extended to an ALE co-ordinate frame. The solid equations are solved via a Jacobi iterative dual-timestepping scheme which is implemented such as to ensure matrix-free and robust solution. Finally, the modelling technology outlined above is validated by application to problems from the literature. The coupled solver is applied to strongly-coupled large-displacement FSI benchmark problems. Rigorous temporal and mesh independent studies are presented.

The outline of this paper is as follows: in Section 2 we present the governing equations for fluid and solid domains, then describe the discretisation, numerical solution and coupling algorithm in Section 3. In Section 4 we detail the mesh movement algorithm used, and in Section 5 discuss parallelisation of the code. We present numerical applications in Section 6 before concluding in Section 7.

2. Governing equations

The fluid–structure interaction to be modelled consists of a viscous incompressible isothermal fluid domain and homogeneous isotropic elastic solid region. The mechanics of each is described via the appropriate governing equation set, which is detailed in this section. Note that for the purposes of this work, the fluid-boundary mesh is fitted to the deforming solid.

2.1. Fluid equations

The fluid flow is governed by the Navier–Stokes equations. In general, these equations are expressed in an Eulerian or spatial frame of reference, which entails a fixed spatial region with fluid flowing through it. For fluid–structure interaction problems the solid deforms and displaces the fluid domain and the fluid equations are written in a manner which allows a Lagrangian description at the fluid–solid interface interpolating to an Eulerian description at the outer boundaries. For this purpose an arbitrary-Lagrangian–Eulerian (ALE) reference frame is used, which accounts for the motion of the FSI interface. The ALE approach was first described by Hirt et al. [46] and later adopted by many others and is now widely used for FSI applications [47]. A dynamic mesh movement algorithm that deforms the fluid mesh is therefore required and is described later. The deforming-spatial-domain/space–time procedure [48,49] is another popular method for treating moving boundaries and interfaces, while other formulations that utilise a fixed mesh, including immersed boundary [50] and fictitious domain [51] methods, can also be used to perform FSI simulations.

Assuming a viscous, incompressible and isothermal fluid, the equations governing the fluid flow are given by the continuity and