Computers and Mathematics with Applications 71 (2016) 869–891

Contents lists available at ScienceDirect

Computers and Mathematics with Applications

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

Parallel coupling numerics for partitioned fluid–structure interaction simulations

Miriam Mehl [a,*], Benjamin Uekermann [b,1], Hester Bijl [c], David Blom [c], Bernhard Gatzhammer [b], Alexander van Zuijlen [c]

[a] Institute for Parallel and Distributed Systems, Universität Stuttgart, Universitätsstraße 38, Stuttgart, Germany
[b] Department of Computer Science, Technische Universität München, Boltzmannstraße 3, Garching, Germany
[c] Faculty of Aerospace Engineering, Delft University of Technology, Kluyverweg 1, 2629HS Delft, The Netherlands

ARTICLE INFO

Article history:
Received 25 May 2015
Received in revised form 18 September 2015
Accepted 14 December 2015
Available online 11 February 2016

Keywords:
Fluid–structure interaction
Partitioned simulation
Parallel coupling methods
Quasi-Newton
High performance computing

ABSTRACT

Within the last decade, very sophisticated numerical methods for the iterative and partitioned solution of fluid–structure interaction problems have been developed that allow for high accuracy and very complex scenarios. The combination of these two aspects – accuracy and complexity – demands very high computational grid resolutions and, thus, high performance computing methods designed for massively parallel hardware architectures. For those architectures, currently used coupling methods, which mainly work with a staggered execution of the fluid and the structure solver, i.e., the execution of one solver after the other in every outer iteration, lead to severe load imbalances: if the flow solver, e.g., scales on a very large number of processors but the structural solver does not due to its limited amount of data and required operations, almost all processors assigned to the coupled simulations are idle during the execution of the structure solver. We propose two new iterative coupling methods that allow for the simultaneous execution of flow and structure solvers. In both cases, we show that pure fixed-point iterations based on the parallel execution of the solvers do not lead to good results, but the combination of parallel solver execution and so-called quasi-Newton methods yields very efficient and robust methods. Those methods are known to be very efficient also for the stabilization of critical scenarios solved with the standard staggered solver execution. We demonstrate the competitive convergence of our methods for various established benchmark scenarios. Both methods are perfectly suited for use with black-box solvers because the quasi-Newton approach uses solely input and output information of the solvers to approximate the effect of the unknown Jacobians that would be required in a standard Newton solver.
© 2016 Elsevier Ltd. All rights reserved.

1. Introduction

Fluid–structure interaction (FSI) simulations are part of a large class of applications involving several physical effects described by different mathematical models, so-called multi-physics applications. With increasing compute power and

* Corresponding author.
E-mail addresses: miriam.mehl@ipvs.uni-stuttgart.de (M. Mehl), uekerman@in.tum.de (B. Uekermann), h.bijl@tudelft.nl (H. Bijl), d.s.blom@tudelft.nl (D. Blom), gatzhamm@in.tum.de (B. Gatzhammer), a.h.vanzuijlen@tudelft.nl (A. van Zuijlen).
[1] Supported by the Institute for Advanced Study, Technische Universität München (www.ias.tum.de).

http://dx.doi.org/10.1016/j.camwa.2015.12.025
0898-1221/© 2016 Elsevier Ltd. All rights reserved.

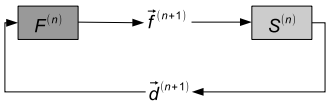

Fig. 1. Standard staggered approach for the coupling of fluid and structure equations: several iterations consisting of a fluid solver step (F) mapping displacements to forces at the wet surface followed by a structural solver step (S) mapping these forces back to displacements are executed per time step. For stabilization purposes, additional coupling numerics (e.g. under relaxation) can be added. We refer to this approach as the Seq-system (staggered or serial).

improved numerical methods, this type of application has become feasible for numerical simulation. Research in multi-physics applications is driven by the need to improve original single-physics models to achieve a higher accuracy. Prominent examples for FSI are the simulation of blood flow in the human cardiovascular system where calcification and aneurysms, and their risk can only be predicted by a combination of flow and structure simulations (e.g. [1]), the weight minimization of airplanes where flow induced vibrations of the structure have an important influence on the stability and durability of the aircraft [2,3], as well as on the produced noise (see, e.g., [4]), or the simulation of opening and descending parachutes (see, e.g., [5–7]).

For many multi-physics applications, highly sophisticated software codes are available for the involved single-physics phenomena. It would be desirable to use them in a modular way for multi-physics simulations. This is particularly true, if an application might later require the addition of further physical phenomena (e.g., fluid–structure-acoustic applications [8]) or the exchange of the single-physics solvers depending on specific needs of the considered scenario. This modular approach to establish a simulation environment is called the partitioned approach (see, e.g., [9]) in contrast to the monolithic approach (see, e.g., [10–12]), where all equations are realized in a single code and, in particular, discretized and solved as a single large system of equations. In addition to the pure notion of partitioned and monolithic coupling methods, many efficient mixed forms have been developed such as the splitting methods proposed in [13] based on a splitting of the fluid–structure system in a fluid-velocity block and a pressure-structure block similar to projection methods known in fluid dynamics. The issue with these approaches in the context of black-box solvers is that they require access to discretization details (in time and space) of the involved solver codes. This holds also for approaches assembling the exact fluid–structure-interface system or its exact Jacobian [14] in order to apply Krylov methods. A recent overview on multi-physics coupling is presented in [15].

It is hardly surprising that the partitioned approach is more involved if we want to design robust solvers that have only limited information at the interface between the codes. For example, information about the discretization of the single-physics solvers (such as used in [16]) is, in general, not available. The solvers act as black-boxes delivering certain physical quantities such as velocities, pressures, forces, and temperatures, but no further insight in their interior details. However, the gain in development speed of the multi-physics simulation environment and the inherent flexibility (cf. [17–20,9,21–23,4]) make the partitioned approach very attractive. In this paper, we focus solely on partitioned coupling methods.

As mentioned, for partitioned black-box coupling, the coupling conditions between flow and structural equations are not part of a large system of equations including fluid equations, structural equations, and coupling. Instead, fluid and structure solvers are considered as closed entities with a given input–output relation at the common boundary, the so-called wet surface. More precisely, the fluid solver usually[2] uses velocities as an input and returns forces as an output. The structure solver takes forces as an input and returns displacements and, sometimes, velocities. For black-box solvers, we cannot assume any further knowledge. This leads to the following simplified view of the action of the fluid and the structural solver at the wet surface:

$$F^{(n)} : \mathbb{R}^{d \cdot M_d} \to \mathbb{R}^{d \cdot M_f}, \quad \vec{d} \mapsto \vec{f},$$
$$S^{(n)} : \mathbb{R}^{d \cdot M_f} \to \mathbb{R}^{d \cdot M_d}, \quad \vec{f} \mapsto \vec{d}$$

where d denotes the dimensionality of our scenario (1, 2 or 3), M_d and M_f are the numbers of grid nodes for the displacements and the forces, respectively, at the wet surface, $\vec{d}$ the displacements, and $\vec{f}$ the forces at the wet surface. $F^{(n)}$ denotes the action of the fluid solver in the time step $t_n \to t_{n+1}$ at the wet surface—the mapping of the input displacements at the wet surface to the forces exerted on the structure. Analogously, $S^{(n)}$ describes the effect of the structural solver's time step $t_n \to t_{n+1}$ on the wet surface values—the mapping of forces to displacements. For some methods described in this work, we have to invert the relation between input and output for the structure solver. Thus, we use $S^{(n),-1}$ to denote a time step of the structure solver using velocities as input and yielding forces as output.

The most widely used method to couple fluids and structures uses a staggered execution order of the respective solvers: a fluid solver step computing forces from velocities, followed by a structural solver step computing velocities from these forces. These two steps are repeated within one time step of the overall simulation if we want to ensure a strong coupling of the involved equations. Fig. 1 schematically shows the algorithm of this standard coupling iteration.

In some cases, this strong coupling is the only way to achieve stability of the resulting time stepping method. The reason is the so-called added-mass effect (cf. [24–26]). Furthermore, it is usually necessary to enhance the described staggered

[2] Usually here means in the context of a Dirichlet–Neumann coupling between flow and structural solver, a very common way of coupling the two systems of equations.

Download English Version:

https://daneshyari.com/en/article/470804

Download Persian Version:

https://daneshyari.com/article/470804

Daneshyari.com