Lecture Notes in Computational Science and Engineering

Fluid-structure interactions (FSI), that is interactions of some movable or deformable structure with an internal or surrounding fluid flow, are among the most important and, with respect to both modelling and computational issues, the most challenging multi-physics problems. The variety of FSI occurrences is abundant and ranges from tent-roofs to micropumps, from parachutes via airbags to blood flow in arteries. This volume of LNCSE contains a collection of papers presented at the International Workshop on FSI held in October 2003 in Hohenwart and organized by DFG's Research Unit 493 "FSI: Modelling, Simulation, and Optimization". The papers address partitioned and monolithic coupling approaches, methodical issues and applications, and discuss FSI from the mathematical, informatical, and engineering point of view.

ISSN 1439-7358
ISBN 3-540-34595-7

springer.com

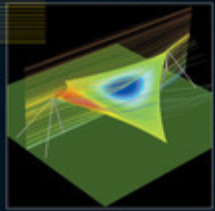
53 LNCSE

H.-J. Bungartz M. Schäfer (Eds.)

Fluid-Structure Interaction

Lecture Notes in Computational Science and Engineering

53



Hans-Joachim Bungartz
Michael Schäfer
Editors

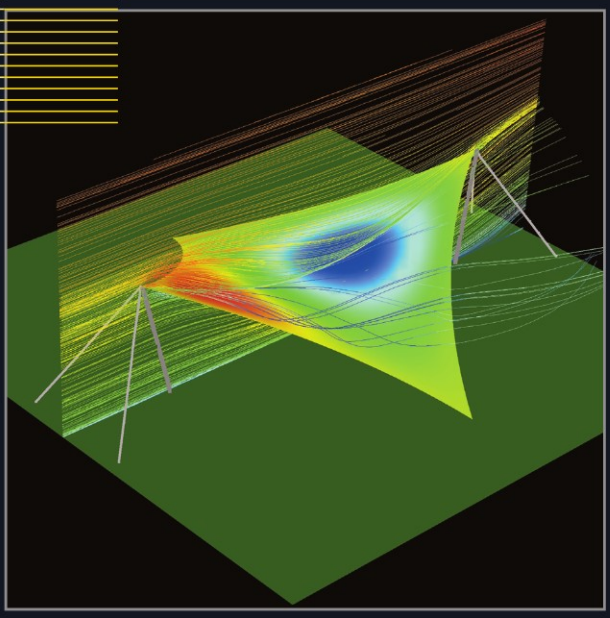
Fluid-Structure Interaction

Modelling, Simulation, Optimization

Springer

Lecture Notes in Computational Science and Engineering

53



Hans-Joachim Bungartz
Michael Schäfer
Editors

Fluid-Structure Interaction

Modelling, Simulation, Optimization

Springer

Lecture Notes
in Computational Science
and Engineering

53

Editors

Hans-Joachim Bungartz Michael Schäfer (Eds.)

Fluid-Structure Interaction

Modelling, Simulation, Optimisation

With 251 Figures and 48 Tables

Springer

Editors

Hans-Joachim Bungartz
Technische Universität München
Institut für Informatik
Boltzmannstraße 3
85748 Garching, Germany
email: bungartz@in.tum.de

Michael Schäfer
Technische Universität Darmstadt
Fachgebiet Numerische Berechnungsverfahren
im Maschinenbau
Petersenstraße 30
64287 Darmstadt, Germany
email: schaefer@fnb.tu-darmstadt.de

Library of Congress Control Number: 2006926465

Mathematics Subject Classification: 65-06, 65Mxx, 65Nxx, 65Y05, 74F10, 74Sxx, 76D05

ISBN-10 3-540-34595-7 Springer Berlin Heidelberg New York
ISBN-13 978-3-540-34595-4 Springer Berlin Heidelberg New York

Springer is a part of Springer Science+Business Media
springer.com

Printed in The Netherlands

Typesetting: by the authors and techbooks using a Springer LaTeX macro package
Cover design: *design & production* GmbH, Heidelberg

Printed on acid-free paper SPIN: 11759751 46/techbooks 5 4 3 2 1 0

Preface

The increasing accuracy requirements in many of today's simulation tasks in science and engineering more and more often entail the need to take into account more than one physical effect. Among the most important and, with respect to both modelling and computational issues, most challenging of such 'multiphysics' problems are fluid-structure interactions (FSI), i.e. interactions of some movable or deformable elastic structure with an internal or surrounding fluid flow. The variety of FSI occurrences is abundant and ranges from huge tent-roofs to tiny micropumps, from parachutes and airbags to blood flow in arteries.

Although a lot of research has been done in this thriving field, with sometimes really impressive results, and although most of today's software packages for computational fluid dynamics or computational structural mechanics offer extensions that, at least to some extent, allow for simulating certain classes of FSI scenarios, some of the key questions have not been answered yet in a satisfying way: How can the coupling itself be modelled in an appropriate way? What are the possibilities and limits of monolithic and partitioned coupling schemes or hybrid approaches? What can be said concerning the advantages and drawbacks of the various discretization schemes used on the flow and on the structure side? How reliable are the results, and what about error estimation? How can a flexible data and geometry model look like – especially against the background of large geometric or even topological changes? What can be said about the design of robust and efficient solvers? And how can sensitivity and optimization issues enter the game?

The book in hand contains the proceedings of a workshop on fluid-structure interactions held in Hohenwart, Germany, in October 2005. This 2-day workshop was organized by the Research Unit 493 'Fluid-Structure Interaction: Modelling, Simulation, Optimization' established by the Deutsche Forschungsgemeinschaft (DFG) in 2003 and bringing together researchers from seven German universities from the fields of mathematics, informatics, mechanical engineering, chemical engineering, and civil engineering. Designed as a forum for presenting the research unit's latest results as well as for exchanging ideas with leading international experts, the workshop consisted of fifteen lectures on computational aspects of fluid-structure interactions. The topics now gathered in this volume cover a broad spectrum of up-to-date FSI issues, ranging from more methodical aspects to applications.

We would like to thank the editors of Springer's Lecture Notes in Computational Science and Engineering for admitting our volume to this series and Springer Verlag and, in particular, Dr. Martin Peters, for their helpful support from the first ideas up to the final layout. Furthermore, we are obliged to Markus Brenk, who did a great job in compiling the single contributions

to a harmonic ensemble. Finally, we are grateful for the Research Unit 493 'Fluid-Structure Interaction: Modelling, Simulation, Optimization' funded by the Deutsche Forschungsgemeinschaft (DFG). Without this financial support, neither many of the results presented in this book nor the book itself would have been possible.

München and Darmstadt *Hans-Joachim Bungartz*
March 2006 *Michael Schäfer*

Table of Contents

Implicit Coupling of Partitioned Fluid-Structure Interaction Solvers using Reduced-Order Models

Jan Vierendeels

Ghent University, Department of Flow, Heat and Combustion Mechanics,
St.-Pietersnieuwstraat 41, B-9000 Ghent, Belgium

Abstract. In this contribution a powerful technique is described which allows the strong coupling of partitioned solvers in fluid-structure interaction (FSI) problems. The method allows the use of a black box fluid and structural solver because it builds up a reduced order model of the fluid and structural problem during the coupling process. Each solution of the fluid/structural solver in the coupling process can be seen as a sensitivity response of an applied displacement/pressure mode. The applied modes and their responses are used to build up the reduced order model. The method is applied on the flow in the left ventricle during the filling and emptying phase. Two to three modes are needed, depending on the moment in the heart cycle, to reduce the residual by four orders of magnitude and to achieve a fully coupled solution at each time step.

1 Introduction

The computation of fluid-structure interaction (FSI) problems has gain a lot of interest in the past decade. The interaction can be loose or strong. For loose coupling problems (e.g. for flutter analysis [1–3]) existing fluid and structural solvers can be used as partitioned solvers. The main difficulty is the data exchange between those solvers.

When strong interaction is present, strong coupling of the fluid and structural solver can be achieved with a monolithic scheme [4]. However partitioned schemes can also be used for these applications. Vierendeels et al. [5,6] used a partitioned procedure and reached stabilization of the interaction procedure by introducing artificial compressibility in the subiterations by preconditioning the fluid solver. Recently strongly coupled partitioned methods were developed [7–10] using approximate or exact Jacobians of the fluid and structural solver. In these methods no black box fluid and/or solid solver can be used.

When existing fluid and structural solvers are used to solve strongly coupled FSI problems, a subiteration process has to be set up for every time step in order to achieve the strong coupling, but in order to obtain convergence typically quite a lot of subiterations are required. Mok et al. [11] used an Aitken-like method to enhance the convergence behaviour of this subiteration process.

In this contribution a coupling procedure is presented which outperforms the Aitken-like method for strongly coupled FSI problems. A partitioned procedure is used and implicit coupling is achieved through sensitivity analysis of the important displacement and pressure modes. These modes are detected during the subiteration procedure for each time step. The method allows the use of black box fluid and structural solver. The method is applied to a 2D axisymmetrical model of the cardiac wall which motion is computed during a complete heart cycle. The structural solver was already developed in previous work [5]. As fluid solver the commercial CFD software package Fluent 6.1 (Fluent Inc.) is used to illustrate the practical applicability of the method.

2 Methods

2.1 Fluid and Structural Solver

The black box fluid solver which is used has to fulfill some conditions. It must be possible to prescribe the movement of the boundary of the fluid domain through e.g. a user subroutine and it must be possible to extract the stress data at the moving boundaries. In our application we only need the pressure distribution at the moving boundary. The response of the flow solver can be represented by the function F:

$$p_{k+1}^{n+1} = F^{n+1}\left(X_{k+1}^{n+1}\right), \tag{1}$$

where X_{k+1}^{n+1} denotes the prescribed position of the boundary nodes obtained from the structural solver in subiteration $k+1$ when computing the solution on time level $n+1$. It is assumed that the solution on time level n is known. The superscript $n+1$ on F denotes other variables in the flow solver that are already known on time level $n+1$, such as in- and outflow boundary conditions. Starting from time level n the pressure distribution on the boundary nodes p_{k+1}^{n+1} can be computed, which is then passed to the structural solver.

The choice of the boundary conditions needs some attention. When the ventricle is filling the fluid domain has only an inlet, no outlet is present. Therefore it is impossible to specify a velocity at the inlet boundary. This would conflict with the change in volume of the ventricle which is already prescribed by the boundary position on the new time level. Moreover, also the pressure field will be undefined upto a constant value if no pressure boundary is specified. Therefore it is necessary to prescribe the pressure at the inflow boundary during the filling phase and at the outflow during the emptying phase.

The structural model which is used was already developed in previous work [5]. The structural equations are given by G:

$$G^{n+1}\left(X_{k+1}^{n+1}, p_k^{n+1}, \Delta p_{k+1}^{n+1}\right) = 0. \tag{2}$$

Since we are dealing with the cardiac cycle the function G^{n+1} incorporates the prescribed time dependency of the structural properties. In our application,

it is assumed that the volume of the ventricle is known as a function of time, therefore the structural solver does not only compute the new position of the boundary nodes, given a pressure distribution at the boundary, but it also computes a pressure shift, Δp_{k+1}^{n+1}, equal for all nodes, so that the volume corresponds with the prescribed volume at that time level. This pressure shift is used to adjust the pressure level in the fluid calculations by adjusting the pressure level of the boundary conditions. In the sequel we denote the structural equations as

$$G^{n+1}\left(X_{k+1}^{n+1}, p_k^{n+1}\right) = 0 \tag{3}$$

for a given pressure input p_k^{n+1} coming from the fluid solver, neglecting the notation for the update of the pressure boundary condition needed in the fluid solver. The structural solver can also be denoted as

$$X_{k+1}^{n+1} = S^{n+1}(p_k^{n+1}). \tag{4}$$

The superscript $n+1$ on F, G and S are dropped from now on. Equation (3) is solved by Newton's method.

2.2 Classical Strong Coupling Methods for Partitioned Solvers

Explicit subiterations within a time step Strong coupling can be obtained by calling the fluid and structural solver subsequently during the calculation of a time step until convergence is obtained. When there is a lot of interaction between both subproblems, this approach can lead to divergence in the subiteration process. When underrelaxation is introduced with a constant underrelaxation parameter, divergence can be avoided but convergence is not really obtained as is illustrated below.

A non-constant underrelaxation parameter can be used to improve the convergence of the subiteration process. The underrelaxation parameter can be obtained with an Aitken-like acceleration method [11] as follows:

$$\omega^k = \frac{(X^k - X^{k-1}) \cdot (R(X^k) - R(X^{k-1}))}{(R(X^k) - R(X^{k-1})) \cdot (R(X^k) - R(X^{k-1}))} \tag{5}$$

where $R(X) = S \circ F(X) - X$. X^{k+1} can be obtained with

$$X^{k+1} = X^k - \omega^k R(X^k). \tag{6}$$

An initial value for ω has to be chosen. We used an initial value of 0.01.

Comparison of the different classical methods If subsequently the structural solver and the fluid solver are called within the subiterations of a time step, divergence is detected. This is shown in Fig. 1 for the first time step of the first heart cycle at the onset of filling. Even when underrelaxation

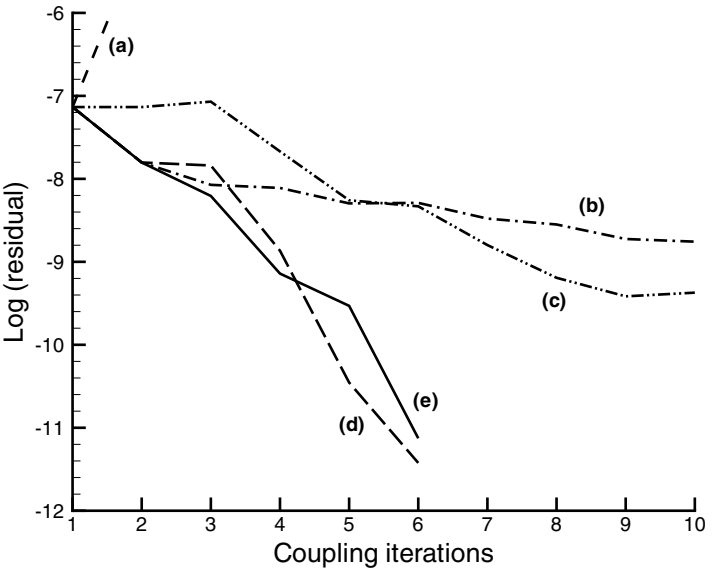


Fig. 1. Residual behaviour of the coupling method for the first time step of the first heart cycle at the onset of filling: (a) no reduced order model, without underrelaxation, (b) no reduced order model, with underrelaxation 0.05, (c) no reduced order model, with Aitken-like acceleration technique, (d) with the reduced order model for the fluid solver, (e) with the reduced order models for both the fluid and structural solver.

is used, convergence within the subiterations could not be obtained in a reasonable number of subiterations (Fig. 1). With the Aitken-like method, convergence was also not really obtained for the first time step within a reasonable number of subiterations. During the next time steps even a worse convergence behaviour was observed.

Figure 2 shows the evolution of the position of the boundary during the subiteration process of the first time step when subsequent calls of strucural and fluid solver without underrelaxation are performed. One can detect that the behaviour of low frequency modes are responsable for the divergence behaviour.

From this observation, it can be expected that when implicitness is introduced in the subiteration process for a few low frequency modes, convergence could be obtained.

2.3 Coupling Method with a Reduced Order Model for the Fluid Solver (Method 1)

Since the fluid solver is a black box commercial code, it is not possible to retrieve or construct the Jacobian F_X, which is needed to solve the structural

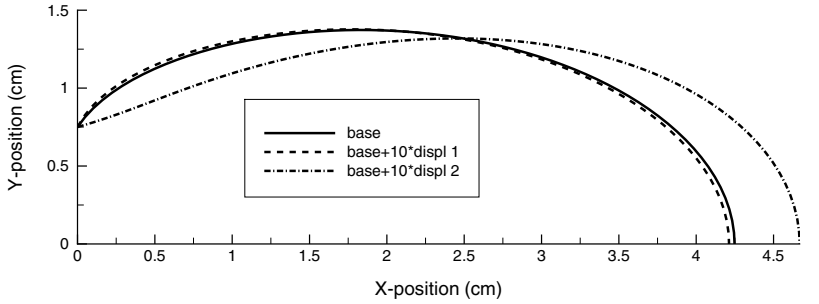


Fig. 2. Illustration of the computed displacements of the heart wall if subsequently the structural and fluid solver are called and when no underrelaxation is used.

problem in an implicit way:

$$G\left(X_{k+1}^{n+1}, p_{k+1}^{n+1}\right) = 0. \tag{7}$$

However it is possible to construct a reduced order model of the fluid solver which can be differentiated easily. Let's denote the reduced order model of the fluid solver by

$$\hat{p}_{k+1}^{n+1} = \hat{F}\left(X_{k+1}^{n+1}\right), \tag{8}$$

then the equations for the structure are written as

$$G\left(X_{k+1}^{n+1}, \hat{p}_{k+1}^{n+1}\right) = 0. \tag{9}$$

A Newton iteration method can be set up after inserting (8) into (9) as follows:

$$G\left(X_{k+1,s}^{n+1}, \hat{p}_{k+1,s}^{n+1}\right) + \left(\frac{\partial G}{\partial X} + \frac{\partial G}{\partial \hat{p}}\frac{\partial \hat{p}}{\partial X}\right)\left(X_{k+1,s+1}^{n+1} - X_{k+1,s}^{n+1}\right) \approx 0, \tag{10}$$

which is solved for $X_{k+1,s+1}^{n+1}$ upon convergence. Remark that this iteration procedure with index s involves only the solution of the structural problem.

The problem can be solved if we have an expression for the Jacobian $\dfrac{\partial \hat{p}}{\partial X}$ of the reduced order model for the fluid problem which we will denote by $\hat{F}_X$ in the sequel.

Construction of the reduced order model After k subiteration loops (and thus k fluid solver calls) k sets of boundary positions and corresponding pressure distributions are obtained that fulfill the flow equations (1). From the moment that minimum two sets $(X_i, p_i), i = 1 \ldots k$ are available, a set of displacement modes $V_m = \{v_m, m = 1 \ldots k-1\}$ is constructed with

$$v_m = X_k - X_m. \tag{11}$$

The corresponding pressure mode to v_m is denoted by $\Delta p_m = p_k - p_m$. A pressure mode matrix ΔP_{k-1} is constructed:

$$\Delta P_{k-1} = \left[\Delta p_1 \cdots \Delta p_{k-1} \right], \tag{12}$$

where the columns contain the computed pressure modes.

An arbitrary displacement ΔX can be projected onto the set of displacement modes V_m. The displacement ΔX can be written as

$$\Delta X = \sum_{m=1}^{k-1} \alpha_m v_m + \Delta X_{corr} \tag{13}$$

where α_m denotes the coordinates of ΔX in the set V_m. Note that the number of displacement modes $(k-1)$ is much smaller than the dimension of ΔX, which explains the correction term. If the displacement modes are well chosen, ΔX can be approximated by $\Delta \tilde{X}$:

$$\Delta X \approx \Delta \tilde{X} = \sum_{m=1}^{k-1} \alpha_m v_m. \tag{14}$$

This is an overdetermined problem for the coordinates α_m, which can be faced with the least square approach. With this approach, the coordinates α_m can be computed as

$$\begin{bmatrix} \alpha_1 \\ \alpha_2 \\ \vdots \\ \alpha_{k-1} \end{bmatrix} = \begin{bmatrix} \langle v_1, v_1 \rangle & \langle v_1, v_2 \rangle & \cdots & \langle v_1, v_{k-1} \rangle \\ \langle v_2, v_1 \rangle & \langle v_2, v_2 \rangle & \cdots & \langle v_2, v_{k-1} \rangle \\ \vdots & \vdots & & \vdots \\ \langle v_{k-1}, v_1 \rangle & \langle v_{k-1}, v_2 \rangle & \cdots & \langle v_{k-1}, v_{k-1} \rangle \end{bmatrix}^{-1} \begin{bmatrix} v_1^T \\ v_2^T \\ \vdots \\ v_{k-1}^T \end{bmatrix} \Delta X \tag{15}$$

The coordinates α_m denote the amount of each mode in the displacement ΔX so that the corresponding change in pressure Δp can be approximated as

$$\Delta p \approx \Delta P_{k-1} \alpha, \tag{16}$$

where $\alpha = [\alpha_1 \cdots \alpha_{k-1}]^T$. The Jacobian $\hat{F}_X$ of the reduced order model can thus be written as

$$\hat{F}_X = \left[\Delta p_1 \cdots \Delta p_{k-1} \right] \begin{bmatrix} \langle v_1, v_1 \rangle & \cdots & \langle v_1, v_{k-1} \rangle \\ \vdots & & \vdots \\ \langle v_{k-1}, v_1 \rangle & \cdots & \langle v_{k-1}, v_{k-1} \rangle \end{bmatrix}^{-1} \begin{bmatrix} v_1^T \\ \vdots \\ v_{k-1}^T \end{bmatrix} \tag{17}$$

The reduced order model, used in subiteration $k+1$ is written as

$$\hat{p}_{k+1}^{n+1} = p_k^{n+1} + \hat{F}_X \left(X_{k+1}^{n+1} - X_k^{n+1} \right). \tag{18}$$

Once eq. (9) is solved for X_{k+1}^{n+1}, p_{k+1}^{n+1} is obtained from eq. (1) and the residual

$$r_{k+1}^{n+1} = G\left(X_{k+1}^{n+1}, p_{k+1}^{n+1} \right) \tag{19}$$

is computed with the pressure from the fluid solver, i.e. not from the reduced order model.