VI International Conference on Computational Methods for Coupled Problems in Science and Engineering
COUPLED PROBLEMS 2015
B. Schrefler, E. Oñate and M. Papadrakakis(Eds)

ON PARALLEL SCALABILITY ASPECTS OF STRONGLY COUPLED PARTITIONED FLUID-STRUCTURE-ACOUSTICS INTERACTION

DAVID S. BLOM*, VERENA KRUPP†, ALEXANDER H. VAN ZUIJLEN*, HARALD KLIMACH†, SABINE ROLLER†, AND HESTER BIJL*

*Delft University of Technology, Faculty of Aerospace Engineering
Kluyverweg 2, 2629HT Delft, The Netherlands
e-mail: d.s.blom@tudelft.nl, web page: http://www.lr.tudelft.nl

†University Siegen, Simulation Techniques & Scientific Computing,
Hölderlinstr. 3, 57076 Siegen, Germany,
e-mail: verena.krupp@uni-siegen.de

Key words: fluid-structure-acoustics interaction, multi-physics, partitioned, parallel

Abstract. Multi-physics simulations, such as fluid-structure-acoustics interaction (FSA), require a high performance computing environment in order to perform the simulation in a reasonable amount of computation time. Currently used coupling methods use a staggered execution of the fluid and solid solver [6], which leads to inherent load imbalances.

In [12] a new coupling scheme based on a quasi-Newton method is proposed for fluid-structure interaction which coupled the fluid and solid solver in parallel. The quasi-Newton method requires approximately the same number of coupling iterations per time step compared to a staggered coupling approach, resulting in a better load balance when running in a parallel environment.

This contribution investigates the scalability limit and load-balancing for a strongly coupled fluid-structure interaction problem, and also for a fluid-structure-acoustics interaction problem. The acoustic far field of the fluid-structure-acoustics interaction problem is loosely coupled with the flow field.

1 INTRODUCTION

Simulation of fluid-structure-acoustics interaction will bring new insight into different applications, as, for example, the sound design of aircraft or wind energy plants. But such applications yield multi-scale problems including different length scales where solving large problems with a monolithic approach would be too expensive.

Also, highly sophisticated software codes are available for each single physical phenomena and it is desirable to reuse currently available codes for multi-physics simulations.

Since different phenomena typically appear spatially separated it is possible to decompose the overall simulation domain into non-overlapping partitions with distinctive treatment and link these partitioned via a surface coupling. This allows the usage of different numerical methods and tailored grid resolution for each individual partitioned. Moreover, each partitions can be solved by individual software codes and we refer to this approach as black-box approach. Additionally, this offers the possibility to benefit from prior experience on how to scale up each single simulation. We are aiming for such flexibility, but the coupling between each pair of physical solvers needs to be carried out carefully, to get a stable overall simulation, while not degrading the scalability.

Whereas for fluid-structure interaction the coupling approach is typically an implicit coupling, which yield same accuracy as a monolithic simulation, for the fluid-acoustic domain with sound generation and propagation several coupling approaches are known. For the flow domain, we are choosing a compressible flow solver, since sound generation implies small perturbations or changes in the density. Far away from the geometry, where only propagation of acoustic waves is relevant, the set of equations can be reduced to the linearized Euler equations. Additionally, the mesh, which needs to be very fine around the geometry, can be much coarser. Another approach would be to use an incompressible flow solver and extract the acoustic sources from the flow as proposed by Lighthill [11]. Note that the acoustic sources need to be extracted and propagated in the flow domain as well.

The immense computational cost of a three-dimensional simulation demands to make efficient use of todays massively parallel supercomputers. A scalable approach which uses the computational resources efficiently and avoid idle processor is indispensable. Load balancing inside one solver is understood very well, but load balancing over individual solvers which are coupled at the boundaries is a different approach.

To achieve such a computational setup, all included components should fulfill the scalability requirements. The high order discontinuous Galerkin solver Ateles, included in the APES framework [14], scales up to more than 100k CPU cores, the limit of currently available super computer facilities. It combines an octree-based data structure with a very local numerical scheme. The used coupling tool preCICE [9], which is responsible for the data exchange as well as for all interpolations between interfaces, is also executed in parallel and minimizes the amount of communication, since it works local on solver processes. OpenFOAM uses a standard domain decomposition approach, since an unstructured mesh is considered. The governing equations are solved in the separate domains and coupled on the boundaries with appropriate boundary conditions.

The presented coupling approach using scalable software allows solving large multi-scale problem on massively parallel machine where the numerical method as well as the computational resources can be perfectly tailored to the physics.

2 SIMULATION SETUP

The computational domain is partitioned into a fluid, structure and acoustics domain. In each domain different governing equations are solved, and also specialized solvers are developed in order to be able to easily switch between different models and software packages. At the fluid-structure interface, and at the fluid-acoustics interface a coupling tool is used to apply the appropriate boundary conditions.

2.1 Fluid solver

The flow is governed by the compressible Navier-Stokes equations. Therefore, the governing equations for the fluid are given by the mass continuity equation

$$\frac{\partial \rho^f}{\partial t} + \nabla \cdot \left(\rho^f \, \boldsymbol{v}\right) = 0, \tag{1}$$

the balance of momentum

$$\frac{\partial \left(\rho^f \, \boldsymbol{v}\right)}{\partial t} + \nabla \cdot \left(\rho^f \, \boldsymbol{v} \, \boldsymbol{v}\right) = -\nabla p^f + \nabla \cdot \boldsymbol{\tau} + \boldsymbol{B}, \tag{2}$$

and the balance of energy equation

$$\frac{\partial}{\partial t}\left(\rho^f \left(e + \frac{1}{2}\boldsymbol{v}\cdot\boldsymbol{v}\right)\right) + \nabla \cdot \left(\rho^f \, \boldsymbol{v} \left(e + \frac{1}{2}\boldsymbol{v}\cdot\boldsymbol{v} + \frac{p^f}{\rho^f}\right)\right) = -\nabla \cdot \boldsymbol{q} + \nabla \cdot \left(\boldsymbol{\tau}\cdot\boldsymbol{v}\right). \tag{3}$$

The velocity field is denoted by $\boldsymbol{v}^f$, the pressure field is denoted with p^f, and the density is given as ρ^f. The viscous stress tensor is given by $\boldsymbol{\tau}$, the internal energy by e, and the heat flux by $\boldsymbol{q}$. The body force is denoted by $\boldsymbol{B}$.

The foam-extend-3.1 package[1] is used for the simulations, a fork of the well known OpenFOAM package[2]. An unstructured mesh is used for the fluid domain, which gives the advantage to easily generate a grid for complex geometries, which is often the case for an engineering fluid-structure interaction test case. A second finite volume discretization is applied in space, and a second order backward differencing time integration scheme is used to propagate the solution in time. Regarding the pressure-velocity coupling, a coupled solution algorithm [4, 3] is employed. The continuity and momentum equations are solved in a fully coupled implicit manner. Thereafter, the energy equations is solved in a segregated manner.

Due to the use of an unstructured mesh, a mesh deformation technique based on radial basis function interpolation is used to interpolate the displacement of the fluid-structure interface into the complete flow field [5].

Coupling an implicit BDF2 time integration scheme for the flow with an explicit second or fourth order Runge Kutta scheme for the acoustics reduces to a first order time

[1] `http://www.extend-project.de/`

[2] `http://www.openfoam.org/`

integration for the acoustic domain. This poses extra restrictions on the used time step in order to have a stable integration.

At the fluid-acoustics interface, an interpolation is performed in order to transfer the pressure, density, and velocities from OpenFOAM to Ateles via the coupling tool preCICE. The ideal situation would be to use conforming or matching grids at the interface. However, due to the coupling of a finite volume solver with a discontinuous Galerkin solver, the meshes are not matching at the interface. Therefore, a radial basis function interpolation is utilized at the interface in order to reduce the introduced numerical errors due to the partitioning as much as possible.

2.2 Structure solver

The configuration of the structure domain is described by the displacement $\boldsymbol{u}^s$. An elastic and compressible structure is assumed, and the governing equation is given by the balance of momentum

$$\rho^s \frac{\partial \boldsymbol{v}^s}{\partial t} + \rho^s \left(\nabla \boldsymbol{v}^s\right) \boldsymbol{v}^s = \nabla \cdot \boldsymbol{\sigma}^s + \rho^s \boldsymbol{g} \qquad \text{in } \Omega^s. \tag{4}$$

Equation (4) is modified to use the total Lagrangian description, i.e. with respect to the initial reference state Γ^s, resulting in

$$\rho^s \frac{\partial^2 \boldsymbol{u}^s}{\partial t^2} = \nabla \cdot \left(J \boldsymbol{\sigma}^s \boldsymbol{F}^{-T}\right) + \rho^s \boldsymbol{g} \qquad \text{in } \Omega^s, \tag{5}$$

where the deformation gradient tensor $\boldsymbol{F}$ is defined as $\boldsymbol{F} = \boldsymbol{I} + \nabla \boldsymbol{u}^s$, and the Jacobian J is the determinant of the deformation gradient tensor $\boldsymbol{F}$. By applying the constitutive law for the St. Venant-Kirchhoff material, the Cauchy stress tensor $\boldsymbol{\sigma}^s$ is found by applying

$$\boldsymbol{\sigma}^s = \frac{1}{J} \boldsymbol{F} \left(\lambda^s \left(\operatorname{tr} \boldsymbol{E}\right) \boldsymbol{I} + 2\mu^s \boldsymbol{E}\right) \boldsymbol{F}^T, \tag{6}$$

with $\boldsymbol{E} = \frac{1}{2}\left(\boldsymbol{F}^T \boldsymbol{F} - \boldsymbol{I}\right)$, and the shear modulus μ^s [1].

The structure solver is also implemented within the foam-extend-3.1 framework for ease of implementation. Therefore, a finite volume discretization is used instead of a finite element approach.

2.3 Acoustics solver

The acoustic phenomena are also governed by the Navier-Stokes equations (1-3). A simplification can be obtained by neglecting the dissipation terms which leads to the inviscid compressible Euler equations. If there are only small changes in the flow, it can be linearized around the base flow. The base flow is denoted by the subscript 0 and the perturbation is denoted with the superscript a. The linearized variables describe the acoustic phenomena in the fluid. Using the assumption of ideal gas

$$p = \rho R T = (\gamma - 1)\left(e - \frac{\rho \mathbf{v} \cdot \mathbf{v}}{2}\right)$$

a relation between pressure p and energy e yields, where R is the ideal gas constant, T the temperature and γ the isentropic coefficient. In the following, we will treat only the primitive variables density ρ, velocity $\boldsymbol{v}$ and pressure p in the acoustic domain. The linearized Euler equations are given by the linearized equation of mass

$$\frac{\partial \rho^a}{\partial t} + \nabla \cdot (\boldsymbol{v}_0 \rho^a + \rho_0 \boldsymbol{v}^a) = 0, \tag{7}$$

the conservation of the velocity perturbation

$$\frac{\partial \boldsymbol{v}^a}{\partial t} + \nabla \cdot \left(\boldsymbol{v}_0 \boldsymbol{v}^a + \frac{1}{\rho_0} p^a \right) = 0 \tag{8}$$

and the conservation of the pressure perturbation

$$\frac{\partial p^a}{\partial t} + \nabla \cdot (\boldsymbol{v}_0 p^a + \gamma \; p_0 \; \boldsymbol{v}^a) = 0. \tag{9}$$

The acoustic partition is simulated using the high order Discontinuous Galerkin solver Ateles which is included in the APES framework. The Discontinuous Galerkin (DG) method is based on a polynomial representation within an element and flux calculation between elements. The choice of the polynomial degree controls the spatial discretization order. By choosing a high degree of the polynomial function a higher order method can be constructed.

A higher order scheme has different advantages. Firstly, it yields low numerical dissipation and dispersion errors, which is advantageous for approximating the wave propagation over long distances in the acoustic far field. Secondly, it shows high convergence rates in case the solution is smooth. Therefore, a high order scheme uses less elements while still maintaining the same accuracy compared to a second order finite volume method. An explicit second order Runge-Kutta method is used to propagate the solution in time.

The APES [14] framework, in which the solver in included, provides additional tools for pre- and post-processing on the basis of the common mesh library TreElM[3]. The TreElM library [10] relies on an octree representation of the mesh and provides the distributed neighborhood search within that mesh. The APES framework is designed to take advantage of the massively parallel systems available in supercomputing today. Using a space-filling curve for the domain composition of the octree mesh, gives hierarchically structured data and maintains locality. This locality can be perfectly exploited by the DG scheme, which is strongly coupled to data within one element and loosely coupled via element boundaries. Hence, the framework and solver are designed to take advantage of the massively parallel systems available in supercomputing today. By free choices of the spatial scheme order and the hybrid parallelism, the solver can be adapted to the executing machine.

[3] `https://bitbucket.org/apesteam/treelm`

2.4 Fluid-structure interaction

With regards to the fluid-structure interaction problem, the fluid solver and solid solver are considered as black boxes as mentioned in the introduction. In other words, only the input and output information is accessible. Note that the used model for the solid domain can be changed without changing the setup of the fluid-structure interaction problem, since only the input and output information from the fluid solver and solid solver is considered to be accessible.

Therefore, at each time step the response of the fluid solver F_f is defined as

$$\boldsymbol{y} = F_f\left(\boldsymbol{x}\right), \tag{10}$$

where $\boldsymbol{x}$ denotes the displacement of the fluid-structure interface, and $\boldsymbol{y}$ denotes the force acting on the fluid-structure interface. The response of the structure solver F_s is consequently defined as

$$\boldsymbol{x} = F_s\left(\boldsymbol{y}\right). \tag{11}$$

Typically, at every time step the fixed point equation

$$\boldsymbol{x} = F_s \circ F_f\left(\boldsymbol{x}\right) \tag{12}$$

must be satisfied, which can also be written as the interface residual $\boldsymbol{R}$

$$\boldsymbol{R}\left(\boldsymbol{x}\right) = F_s \circ F_f\left(\boldsymbol{x}\right) - \boldsymbol{x}, \tag{13}$$

which is solved with a minimization or optimization procedure aimed to find the optimal solution $\boldsymbol{x}^\star$ such that

$$\boldsymbol{x}^\star = \arg\min_{\boldsymbol{x}} \left|\left|\boldsymbol{R}\left(\boldsymbol{x}\right)\right|\right|_2. \tag{14}$$

At the fluid-structure interaction $\Gamma^{f\,s}$, the balance of stresses is enforced through

$$\boldsymbol{\sigma}^f \boldsymbol{n} = \boldsymbol{\sigma}^s \boldsymbol{n} \qquad \text{on } \Gamma^{f\,s}, \tag{15}$$

with the unit vector $\boldsymbol{n}$ normal to the fluid-structure interface $\Gamma^{f\,s}$, and the stress tensors $\boldsymbol{\sigma}^f$ and $\boldsymbol{\sigma}^s$. Also, the no-slip condition is imposed at the fluid-structure interface so that the velocities $\boldsymbol{v}^f$ and $\boldsymbol{v}^s$ must be equal:

$$\boldsymbol{v}^f = \boldsymbol{v}^s \qquad \text{on } \Gamma^{f\,s}. \tag{16}$$

Standard approaches used to solve the strongly coupled fluid-structure interaction problem are the Gauss-Seidel method [8], fixed under-relaxation [2], Aitken under-relaxation [13], and the IQN-ILS method [6, 7]. Here, the IQN-ILS method is used to solve the interface problem.

In case a compressible flow is simulated, an explicit coupling at the fluid-structure interface can also be applied. However, in this contribution the use of an implicit coupling

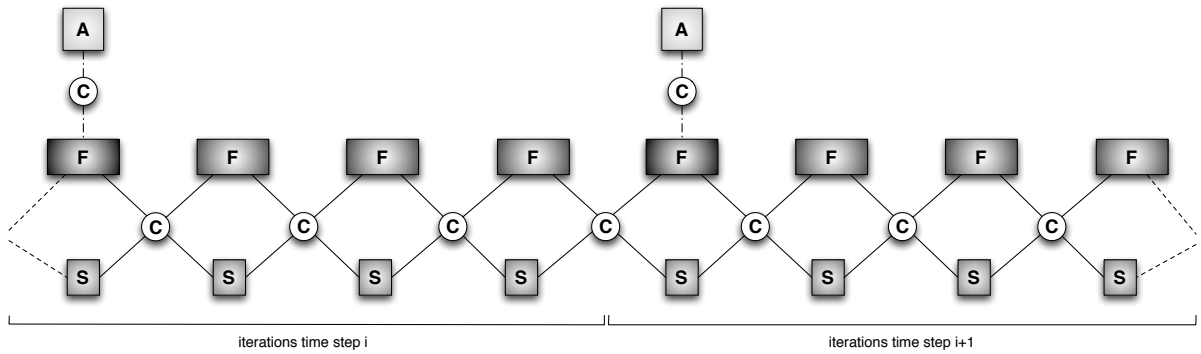

Figure 1: Overview of the execution of the fluid-structure-acoustics simulations. Multiple calls to the fluid and solid solvers are performed, since an implicit coupling is applied for the fluid-structure interaction problem. A good load balancing can be achieved with the proper number of cores for the acoustic domain. F: fluid, S: solid, A: acoustics.

technique is applied in order to remove the introduced partitioning error at the fluid-structure interface.

A distinction is made between the case where the solvers are coupled sequentially, as is generally the case, and in parallel. In [12] a new coupling scheme based on the IQN-ILS method is proposed which coupled the fluid and solid solver in parallel. Hence, a better load balancing is obtained giving the possibility to be able to scale a partitioned fluid-structure interaction further compared to the serially coupled IQN-ILS algorithm.

2.5 Fluid-structure-acoustics interaction

The coupling to the acoustic field is treated with an explicit coupling. In other words, in every time step the primitive variables (pressure, density, velocity) present at the fluid-acoustics interface are transferred via the coupling tool and applied with a boundary condition to the acoustic domain. Thereafter, the linearized Euler equations are solved in the acoustic field.

An overview of the used coupling technique is shown in Figure 1. As shown in the figure, the fluid and solid solvers are executed multiple times per time step. When the FSI iterations are converged, the acoustic solver is called. An optimal load balancing can be obtained by selecting the appropriate number of CPU cores for the acoustic domain, such that the computational time is approximately equal to the time required for the FSI iterations to converge.

3 NUMERICAL RESULTS

The used test case consists of a flexible cantilever in a cross flow. At the outer boundaries the fluid domain is coupled with the acoustic domain. Whereas are th structure-fluid interface is matching, the fluid-acoustic interface differs. In contrast to the fluid domain, which has 124 elements at the interface, the acoustic domain has only 24 elements. Hence,

Table 1: Parallel scalability test for a fluid-structure-acoustics simulation. The fluid, solid and acoustics solvers are coupled in a staggered sense, which leads to an non-optimal load balancing since processors of 2 solvers are always idle, which hurts particularly for the flow solver. The number of fluid cores varies for the computations shown here. The solid and acoustics solver both use one CPU core. 29 time steps are simulated.

No. fluid cores	Exec. time [s]	Timing fluid [s]	Timing solid [s]	Timing acoustics [s]
32	225	187.33	19.37	33.25
64	126	93.74	18.62	35.43
128	80	45.98	18.74	30.99
256	60	22.6	19.75	33.88

Table 2: Parallel scalability tests for a fluid-structure-acoustics simulation. The fluid, solid and acoustics solvers are coupled in parallel, leading to good load-balancing compared to the staggered coupling test case. The number of fluid cores varies for the computations shown here. The solid and acoustics solver both use one CPU core. 29 time steps are simulated.

No. fluid cores	Exec. time [s]	Timing fluid [s]	Timing solid [s]	Timing acoustics [s]
32	225	199.03	20.32	34.94
64	124	99.09	21.77	33.60
128	69	47.97	21.65	33.55
256	47	24.16	20.93	34.87

a fifth order discretization in space is used for the acoustic domain. The pressure field of the test case at $t = 0.2\,s$ is shown in Figure 2.

Table 1 and 2 show the scalability results comparing the use of a serial and parallel coupling scheme for the complete fluid-structure-acoustics interaction setup. The number of cores for the structure and acoustics solvers are kept at one. The number of cores for the fluid is increased from 32 to 256.

The results show the overhead of the communication between the different solvers, and show the efficiency of the parallel coupling schemes. With a proper load balancing, if the solvers are executed in parallel, smaller execution times are obtained in comparison to sequentially coupled solvers. More importantly, this parallel coupling combined with balanced work-load avoid idle processor and can make efficient use of today's supercomputers.

4 CONCLUSIONS

The used methodology has been described for a partitioned fluid-structure-acoustics setup. The fluid-structure interaction problem is coupled implicitly, whereas the fluid and acoustics domain are coupled explicitly. Scaling tests have been performed in order to examine the bottleneck of this approach. The use of a separate coupling tool between the different domains imposes extra communication costs. Further research is dedicated to use a asynchronous communication between the different solver. Also, the use of a

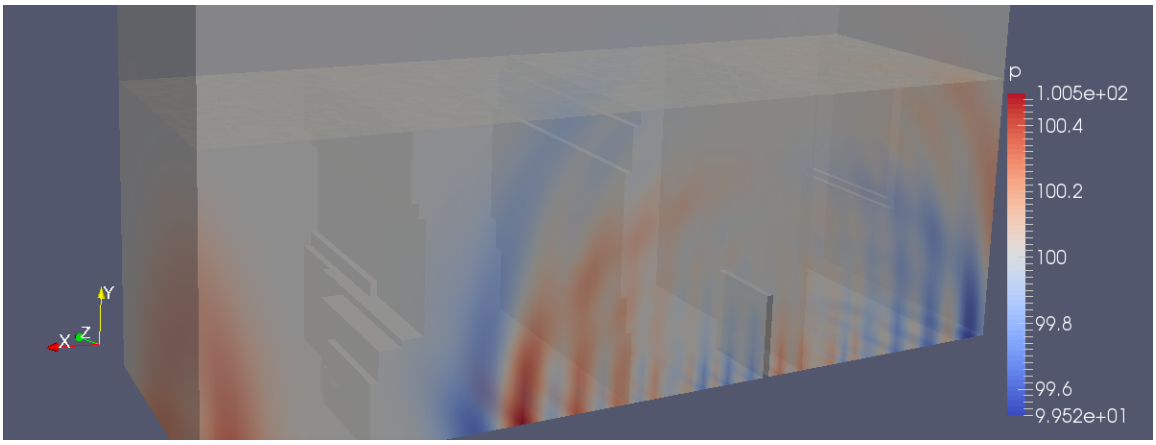

Figure 2: Pressure field for the fluid-structure-acoustics domain at t = 0.2 s. The domain decomposition of the flow is visible in the background. The fluid domain consists of 198 112 cells, the solid domain contains 288 cells, and the acoustic domain consists of 6 144 elements.

spectral deferred correction time integration scheme is considered in order to solve the complete fluid-structure-acoustics domain with a high order time integration scheme.

REFERENCES

[1] P. Cardiff, A. Karač, and A. Ivanković. A large strain finite volume method for orthotropic bodies with general material orientations. *Computer Methods in Applied Mechanics and Engineering*, 268:318–335, January 2014.

[2] M. Cervera, R. Codina, and M. Galindo. On the computational efficiency and implementation of block-iterative algorithms for nonlinear coupled problems. *Engineering Computations*, 13(6):4–30, 1996.

[3] M. Darwish and F. Moukalled. A Fully Coupled Navier-Stokes Solver for Fluid Flow at All Speeds. *Numerical Heat Transfer*, 65(5), 2014.

[4] M. Darwish, I. Sraj, and F. Moukalled. A coupled finite volume solver for the solution of incompressible flows on unstructured grids. *Journal of Computational Physics*, 228(1):180–201, 2009.

[5] A. de Boer, A. H. van Zuijlen, and H. Bijl. *Radial Basis Functions for Interface Interpolation and Mesh deformation*, volume 71 of *Lecture Notes in Computational Science and Engineering*, chapter 6, pages 143–178. Springer Berlin Heidelberg, 2010.

[6] J. Degroote, K.-J. Bathe, and J. Vierendeels. Performance of a new partitioned procedure versus a monolithic procedure in fluid-structure interaction. *Computers and Structures*, 87(11-12):793–801, June 2009.

[7] J. Degroote and J. Vierendeels. Multi-level quasi-Newton coupling algorithms for the partitioned simulation of fluid-structure interaction. *Computer Methods in Applied Mechanics and Engineering*, 225-228:14–27, 2012.

[8] C. Farhat and M. Lesoinne. Two efficient staggered algorithms for the serial and parallel solution of three-dimensional nonlinear transient aeroelastic problems. *Computer Methods in Applied Mechanics and Engineering*, 182(3-4):499–515, February 2000.

[9] B. Gatzhammer, M. Mehl, and T. Weinzierl. A coupling tool for the partitioned simulation of fluid-structure interactions. In T. Kvamsdal, B. Pettersen, P. Bergan, E. Onate, and J. Garcia, editors, *Computational Methods in Marine Engineering*, volume 3, pages 147–150, Trondheim, June 2009. International Center for Numerical Methods in Engineering (CIMNE), CIMNE.

[10] H. Klimach, M. Hasert, J. Zudrop, and S. Roller. Distributed Octree Mesh Infrastructure for Flow Simulations. *European Congress on Computational Methods in Applied Sciences and Engineering*, pages 1–15, 2012.

[11] M.J. Lighthill. On Sound Generated Aerodynamically. I. General Theorie. *Proceedings of The Royal Society London A*, 211:564–587, 1952.

[12] M. Mehl, B. Uekermann, H. Bijl, D. S. Blom, B. Gatzhammer, and A. H. van Zuijlen. Parallel coupling numerics for partitioned fluid-structure interaction simulations. *SIAM Scientific Computing*, submitted 2013.

[13] D. P. Mok, W. A. Wall, and E. Ramm. Accelerated iterative substructure schemes for instationary fluid-structure interaction. In *First MIT Conference on Computational Fluid and Solid Mechanics*, pages 1325–1328, 2001.

[14] J. Zudrop, H. Klimach, M. Hasert, K. Masilamani, and S. Roller. A fully distributed CFD framework for massively parallel systems. In *Cray User Group 2012*, Stuttgart, Germany, April 2012.