A cut-cell finite volume – finite element coupling approach for fluid-structure interaction in compressible flow

Vito Pasquariello^{1*}, Georg Hammerl², Felix Örley¹, Stefan Hickel^{1,3}, Caroline Danowski², Alexander Popp², Wolfgang A. Wall², Nikolaus A. Adams¹

¹Institute of Aerodynamics and Fluid Mechanics, Technische Universität München Boltzmannstr. 15, 85748 Garching bei München, Germany

²Institute for Computational Mechanics, Technische Universität München Boltzmannstr. 15, 85748 Garching bei München, Germany

³Aerodynamics Group, Faculty of Aerospace Engineering, Technische Universiteit Delft Kluyverweg 1, 2629 HS Delft, The Netherlands

Abstract

We present a loosely coupled approach for the solution of fluid-structure interaction problems between a compressible flow and a deformable structure. The method is based on staggered Dirichlet-Neumann partitioning. The interface motion in the Eulerian frame is accounted for by a conservative cut-cell Immersed Boundary method. The present approach enables subcell resolution by considering individual cut-elements within a single fluid cell, which guarantees an accurate representation of the time-varying solid interface. The cut-cell procedure inevitably leads to non-matching interfaces, demanding for a special treatment. A Mortar method is chosen in order to obtain a conservative and consistent load transfer. We validate our method by investigating two-dimensional test cases comprising a shock-loaded rigid cylinder and a deformable panel. Moreover, the aeroelastic instability of a thin plate structure is studied with a focus on the prediction of flutter onset. Finally, we propose a three-dimensional fluid-structure interaction test case of a flexible inflated thin shell interacting with a shock wave involving large

Preprint submitted to Journal of Computational Physics

^{*}Corresponding author. Tel.: +49 89 289 16134; fax: +49 89 289 16139 Email address: vito.pasquariello@tum.de (Vito Pasquariello¹)

and complex structural deformations.

Keywords: Fluid-structure interaction, Compressible flow, Cut-cell method, Immersed boundary method, Mortar method

1. Introduction

Compressible fluid-structure interaction (FSI) occurs in a broad range of technical applications involving, e.g., nonlinear aeroelasticity [16, 42] and shock-induced deformations of rocket nozzles [23, 55]. The numerical modeling and simulation of compressible FSI can be challenging, in particular if an accurate representation of the structural interface within the fluid solver and a consistent coupling of both subdomains is required.

FSI algorithms are generally classified as monolithic or partitioned. One main advantage often attributed to monolithic approaches is their numerical robustness due to solving a single system which includes the full information of the coupled nonlinear FSI problem. On the other hand, partitioned algorithms for FSI are often used because they facilitate the coupling of different specialized single-field solvers. A further distinction can be made between loosely and strongly coupled algorithms, depending on whether the coupling conditions are satisfied exactly at each time step, or not. While partitioned algorithms can be made strong by introducing equilibrium iterations [34], loosely coupled approaches are more frequently used in the field of aeroelasticity and compressible flows in general [6, 16]. A disadvantage of loosely coupled partitioned algorithms is the artificial added mass effect [5, 21], which may lead to numerical instability in incompressible flows and for high fluid-solid density ratios. Recently, so-called Added-Mass Partitioned algorithms have been developed for compressible fluids interacting with rigid and elastic solids [1, 3] as well as for incompressible fluids [2]. These methods allow to overcome the added mass instability by formulating appropriate fluid-structure interface conditions.

FSI problems involve a load and motion transfer at the conjoined interface. In the simple case of matching fluid and solid discretization, this results in a trivial task. However, different resolution requirements within the fluid and solid fields lead to non-matching discrete interfaces. An overview of existing coupling methods for non-matching meshes can be found in [9]. Simple methods such as nearest-neighbor interpolation and projection methods are frequently used [17, 31]. The mentioned methods do not conserve angular momentum across the interface. Consistency can be achieved with more sophisticated approaches, such as weighted residual methods, which introduce Lagrange multipliers as additional interface variables. In this context, Mortar methods have first been proposed for non-overlapping domain decomposition in [4], enhanced with dual shape functions for the Lagrange multipliers in [53] and applied to FSI problems and mesh tying in fluid flow, e.g. in [13, 33]. While Mortar methods introduce Lagrange multipliers only on one side of the interface, Localized Lagrange Multipliers consider them on both sides of the interface [47].

Another classification of FSI methods is based on the representation of the time-varying solid interface within the fluid domain. Two main approaches can be distinguished in this context, which are Arbitrary Lagrangian Eulerian (ALE) methods [10, 18], and Immersed Boundary Methods (IBM) [38, 41]. ALE approaches employ body-fitted grids, hence requiring a mesh evolution algorithm. This task may be complex in case of large solid displacements. On the other hand, IBM often operate on fixed Cartesian fluid grids, making this type of approach very appealing for the simulation of flows past complex geometries and for the solution of FSI problems with large deformations. IBM, such as continuous forcing and ghost-cell approaches, may suffer from spurious loss or production of mass, momentum and energy at the interface [38]. Such non-conservativity poses a particular problem for large-eddy simulations, which employ coarse grids and rely on an accurate flow prediction in near-wall regions over large time scales. Moreover, the accurate capturing of shocks is based on conservation properties. Conservativity is recovered with Cartesian cut-cell methods, which were first introduced by Clarke et al. [7] and Gaffney and Hassan [22] for inviscid flows and later extended to viscous flows by Udaykumar et al. [52] and Ye et al. [54]. In this method, the finite volume cells at the boundaries are reshaped to fit locally the boundary surface with a sharp interface, which in turn assures strict conservation of mass, momentum and energy. A drawback of cut-cell methods is that the fluid volume fraction of cut-cells may become very small and therefore can lead to numerical instability with explicit time integration schemes. A stabilization of the underlying time integration scheme can be achieved by so-called cell-merging [54], cell-linking [32] or flux redistribution techniques [8, 30].

In this paper we develop a loosely coupled approach for the solution of FSI problems between a compressible fluid and a deformable structure. We employ the Finite Volume Method (FVM) for solving the Euler equations on Cartesian grids and the Finite Element Method (FEM) for solving the structural problem. The interface motion is accounted for by a conservative cut-cell IBM. Previous proposed methods reconstruct the interface geometry based on a level-set function [26, 27, 36]. Örley et al. [40] developed a conservative cut-element method that allows for representing the fluid-solid interface with sub-cell resolution for rigid body motion. We extend this method to arbitrary interface deformations. The combination of a cut-element IBM with a Mortar method for coupling of the solid and fluid subdomains in a consistent and efficient way is the essential new contribution of this paper.

This paper is structured as follows: First, the governing equations for fluid and solid and the fluid-structure interface conditions are introduced in Section 2. Section 3 gives a detailed overview on the numerical treatment of moving boundaries together with the discretization methods used for the fluid. The FEM used to solve the structural problem is presented in Section 4. In Section 5, the staggered coupling algorithm is presented together with the new coupling approach for non-matching interfaces. In Section 6, the method is validated with well-established two-dimensional test cases and a convergence study is presented. In Section 7, we propose a new test case for the interaction between a flexible inflated thin shell and a shock wave, demonstrating in particular the capability of our FSI approach to handle large three-dimensional deformations. Concluding remarks are given in Section 8.

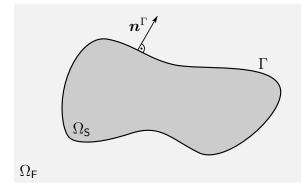


Fig. 1: Schematic of FSI domain.

2. Mathematical and physical model

As depicted in Fig. 1, the computational domain is divided into a fluid and solid domain, Ω_{F} and Ω_{S} , respectively. The conjoined interface is denoted as $\Gamma = \Omega_{\mathsf{F}} \cap \Omega_{\mathsf{S}}$ and its normal vector \boldsymbol{n}^{Γ} in spatial configuration points from the solid into the fluid domain.

2.1. Governing equations for the fluid

We consider the three-dimensional, fully compressible Euler equations in conservative form

$$\frac{\partial \boldsymbol{w}}{\partial t} + \nabla \cdot \boldsymbol{K}(\boldsymbol{w}) = 0 \text{ in } \Omega_{\mathsf{F}} \,. \tag{1}$$

The state vector $\boldsymbol{w} = [\rho_{\mathsf{F}}, \rho_{\mathsf{F}}u_1, \rho_{\mathsf{F}}u_2, \rho_{\mathsf{F}}u_3, E_{\mathsf{t}}]$ contains the conserved variables density ρ_{F} , momentum $\rho_{\mathsf{F}}\boldsymbol{u}$ and total energy E_{t} . The subscript F denotes fluid quantities and is used whenever a distinction between both subdomains is necessary. The individual contributions of the flux tensor $\boldsymbol{K} = (\boldsymbol{f}, \boldsymbol{g}, \boldsymbol{h})$ are given as

$$\boldsymbol{f}(\boldsymbol{w}) = \begin{pmatrix} \rho_{\mathsf{F}} u_{1} \\ \rho_{\mathsf{F}} u_{1}^{2} + p \\ \rho_{\mathsf{F}} u_{1} u_{2} \\ \rho_{\mathsf{F}} u_{1} u_{3} \\ u_{1}(E_{\mathsf{t}} + p) \end{pmatrix}, \, \boldsymbol{g}(\boldsymbol{w}) = \begin{pmatrix} \rho_{\mathsf{F}} u_{2} \\ \rho_{\mathsf{F}} u_{2} u_{1} \\ \rho_{\mathsf{F}} u_{2}^{2} + p \\ \rho_{\mathsf{F}} u_{2} u_{3} \\ u_{2}(E_{\mathsf{t}} + p) \end{pmatrix}, \, \boldsymbol{h}(\boldsymbol{w}) = \begin{pmatrix} \rho_{\mathsf{F}} u_{3} \\ \rho_{\mathsf{F}} u_{3} u_{1} \\ \rho_{\mathsf{F}} u_{3} u_{2} \\ \rho_{\mathsf{F}} u_{3}^{2} + p \\ u_{3}(E_{\mathsf{t}} + p) \end{pmatrix},$$
(2)

where p is the static pressure. We consider a perfect gas with a specific heat ratio of $\gamma = 1.4$ and specific gas constant of $R = 287.058 \frac{J}{kg\cdot K}$. The total

energy is given by

$$E_{\mathsf{t}} = \frac{1}{\gamma - 1} p + \frac{1}{2} \rho_{\mathsf{F}} u_i u_i \,, \tag{3}$$

assuming an ideal gas equation of state $p = \rho_{\mathsf{F}} \mathsf{R} T$, where T is the static temperature. If not stated otherwise, we use the Einstein summation convention.

2.2. Governing equations for the solid

The structural field is governed by the local form of the balance of linear momentum

$$\rho_{\mathsf{S};0}\,\hat{\boldsymbol{d}}\,=\,\nabla_0\cdot\,(\boldsymbol{F}\cdot\boldsymbol{S})\,+\,\hat{\boldsymbol{b}}_0\,\mathrm{in}\,\,\Omega_{\mathsf{S}}\,,\tag{4}$$

describing equilibrium of the forces of inertia, internal and external forces in the undeformed structural domain Ω_{S} . Herein $\nabla_0 \cdot (\bullet)$ is the material divergence operator and the index S represents the domain of the structural problem. The structural material density is denoted by $\rho_{\mathsf{S};0}$. Furthermore, d and \ddot{d} are the unknown displacements and accelerations, respectively. The vector field \hat{b}_0 is the given material body force. The internal forces are expressed in terms of the second Piola-Kirchhoff stress tensor S and the deformation gradient F.

To determine the stresses, various constitutive laws can be used. For the sake of simplicity, in this work a hyperelastic Saint Venant-Kirchhoff material model with strain energy density function Ψ per unit reference volume is chosen as

$$\Psi(\boldsymbol{E}) = \mu_{\mathsf{S}} \boldsymbol{E} : \boldsymbol{E} + \frac{1}{2} \lambda_{\mathsf{S}} (\boldsymbol{E} : \boldsymbol{I})^2, \qquad (5)$$

with the Lamé constants λ_{S} and μ_{S} and the second-order identity tensor I. The Green-Lagrange strain tensor is defined as

$$\boldsymbol{E} = \frac{1}{2} \left(\boldsymbol{F}^{\mathsf{T}} \cdot \boldsymbol{F} - \boldsymbol{I} \right).$$
 (6)

The second Piola-Kirchhoff stress

$$\boldsymbol{S} = \frac{\partial \Psi}{\partial \boldsymbol{E}} \tag{7}$$

is derived using (5). Alternatively, the first Piola-Kirchhoff stress tensor

$$\boldsymbol{P} = \boldsymbol{F} \cdot \boldsymbol{S} \tag{8}$$

may be used.

The boundary of the structural field $\partial \Omega_{S}$ is divided into pairwise disjoint boundary segments

$$\partial\Omega_{\mathsf{S}} = \Gamma_{\mathsf{S};\mathsf{D}} \cup \Gamma_{\mathsf{S};\mathsf{N}} \cup \Gamma \,. \tag{9}$$

On the Dirichlet boundary $\Gamma_{S;D}$, the displacements are prescribed, whereas on the Neumann boundary $\Gamma_{S;N}$, the traction vector \hat{t}_0 is prescribed using the unit normal vector n_0 in material configuration. Thus, the boundary conditions

$$\boldsymbol{d} = \hat{\boldsymbol{d}} \quad \text{on } \Gamma_{\mathsf{S};\mathsf{D}}, \tag{10}$$

$$\boldsymbol{P} \cdot \boldsymbol{n}_0 = \hat{\boldsymbol{t}}_0 \quad \text{on } \Gamma_{\mathsf{S};\mathsf{N}} \tag{11}$$

need to be satisfied.

For the balance equation (4) initial conditions for displacements d and velocities \dot{d} need to be specified at time t = 0,

$$\boldsymbol{d}_0 = \boldsymbol{d}(\boldsymbol{X}, t = 0) = \hat{\boldsymbol{d}}_0 \quad \text{on } \Omega_{\mathsf{S}}, \qquad (12)$$

$$\dot{\boldsymbol{d}}_0 = \dot{\boldsymbol{d}}(\boldsymbol{X}, t = 0) = \dot{\boldsymbol{d}}_0 \quad \text{on } \Omega_{\mathsf{S}}, \qquad (13)$$

where \boldsymbol{X} defines the initial position.

2.3. Fluid-structure interface conditions

Dynamic and kinematic coupling conditions at the conjoined interface Γ ensure the integrity between the subdomains in this partitioned coupling algorithm. Assuming no mass transport across the interface, normal velocities have to match, i.e.

$$\boldsymbol{u}^{\Gamma} \cdot \boldsymbol{n}^{\Gamma} = \frac{\partial \boldsymbol{d}^{\Gamma}}{\partial t} \cdot \boldsymbol{n}^{\Gamma} \quad \text{on } \Gamma, \qquad (14)$$

where \boldsymbol{n}^{Γ} denotes the interface unit normal vector. The dynamic condition requires the tractions to be equal,

$$\boldsymbol{\sigma}_{\mathsf{F}}^{\Gamma} \cdot \boldsymbol{n}^{\Gamma} = \boldsymbol{\sigma}_{\mathsf{S}}^{\Gamma} \cdot \boldsymbol{n}^{\Gamma} \quad \text{on } \Gamma, \qquad (15)$$

where $\sigma_{\mathsf{F}} = -p I$ denotes the fluid stress tensor comprising only contributions due to the pressure in the inviscid case considered here. The Cauchy stress tensor σ_{S} is defined as

$$\boldsymbol{\sigma}_{\mathsf{S}} = \frac{1}{J} \boldsymbol{P} \cdot \boldsymbol{F}^{\mathsf{T}}$$
(16)

in which J is the Jacobian.

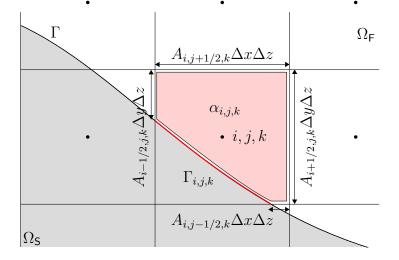


Fig. 2: Two-dimensional sketch of a cut-cell (i, j, k) [40].

3. Numerical approach: Fluid

We employ the FVM for solving the Euler equations on Cartesian grids. The time-dependent fluid-solid interface conditions on Γ are imposed by a cut-element based IBM.

3.1. Mathematical model

A sketch of a two-dimensional cut-cell is shown in Fig. 2. In the following, Γ denotes the fluid-structure interface of the continuous problem, and $\Gamma_{\mathsf{F}/\mathsf{S}}$ the flow and structure side of the interface of the discrete problem. We solve the integral form of (1),

$$\int_{t^n}^{t^{n+1}} \int_{\Omega_{i,j,k} \cap \Omega_{\mathsf{F}}} \left(\frac{\partial \boldsymbol{w}}{\partial t} + \nabla \cdot \boldsymbol{K}(\boldsymbol{w}) \right) \mathrm{d}x \mathrm{d}y \mathrm{d}z \, \mathrm{d}t = 0, \tag{17}$$

where the integral is taken over the volume $\Omega_{i,j,k} \cap \Omega_{\mathsf{F}}$ of a computational cell (i, j, k) and time step $\Delta t = t^{n+1} - t^n$. Applying the Gauss theorem results in

$$\int_{t^n}^{t^{n+1}} \int_{\Omega_{i,j,k} \cap \Omega_{\mathsf{F}}} \frac{\partial \boldsymbol{w}}{\partial t} \, \mathrm{d}V \, \mathrm{d}t + \int_{t^n}^{t^{n+1}} \int_{\partial(\Omega_{i,j,k} \cap \Omega_{\mathsf{F}})} \boldsymbol{K}(\boldsymbol{w}) \cdot \boldsymbol{n} \, \mathrm{d}S \, \mathrm{d}t = 0, \quad (18)$$

where $\partial(\Omega_{i,j,k} \cap \Omega_{\mathsf{F}})$ denotes the wetted surface of a computational cell (i, j, k), and dV, dS the infinitesimal volume and surface element, respectively. Applying a volume average of the conserved variables

$$\mathbf{w}_{i,j,k} = \frac{1}{\alpha_{i,j,k} V_{i,j,k}} \int_{\Omega_{i,j,k} \cap \Omega_{\mathsf{F}}} \boldsymbol{w} \, \mathrm{d}x \mathrm{d}y \mathrm{d}z, \tag{19}$$

leads to

$$\alpha_{i,j,k}^{n+1} \mathbf{w}_{i,j,k}^{n+1} = \alpha_{i,j,k}^{n} \mathbf{w}_{i,j,k}^{n} \\
+ \frac{\Delta t}{\Delta x_{i}} \left[A_{i-1/2,j,k}^{n} \mathbf{f}_{i-1/2,j,k} - A_{i+1/2,j,k}^{n} \mathbf{f}_{i+1/2,j,k} \right] \\
+ \frac{\Delta t}{\Delta y_{j}} \left[A_{i,j-1/2,k}^{n} \mathbf{g}_{i,j-1/2,k} - A_{i,j+1/2,k}^{n} \mathbf{g}_{i,j+1/2,k} \right] \\
+ \frac{\Delta t}{\Delta z_{k}} \left[A_{i,j,k-1/2}^{n} \mathbf{h}_{i,j,k-1/2} - A_{i,j,k+1/2}^{n} \mathbf{h}_{i,j,k+1/2} \right] \\
+ \frac{\Delta t}{V_{i,j,k}} \boldsymbol{\chi}_{i,j,k}.$$
(20)

 $V_{i,j,k} = \Delta x_i \Delta y_j \Delta z_k$ corresponds to the total volume of cell $\Omega_{i,j,k}$, $\alpha_{i,j,k}$ corresponds to the fluid volume fraction, $\mathbf{w}_{i,j,k}$ is the vector of volume-averaged conserved quantities in the cut-cell, and A is the effective fluid wetted cell face aperture. The face averaged numerical fluxes across the cell faces are denoted as \mathbf{f}, \mathbf{g} and \mathbf{h} . The flux $\boldsymbol{\chi}_{i,j,k}$ across the interface $\Gamma_{i,j,k} = \Gamma \cap \Omega_{i,j,k}$ is discussed in detail below.

Time integration of the state vector is shown here for a forward Euler time integration scheme with a time step Δt , which corresponds to one substep of an explicit Runge-Kutta method. Appropriate initial and boundary conditions are prescribed on the domain $\Omega_{\rm F}$ and the surface $\partial \Omega_{\rm F}$. For all simulations presented in this paper we employ a spatial flux discretization on local characteristics by an 5th-order WENO scheme [35] together with a Lax-Friedrichs flux function. A 3rd-order strongly stable Runge-Kutta scheme [25] is used for time integration.

3.2. Conservative immersed boundary method

3.2.1. Geometry computation

Moving boundaries with sharp corners and complex geometries may cause numerical artifacts in terms of spurious pressure oscillations. Following Örley

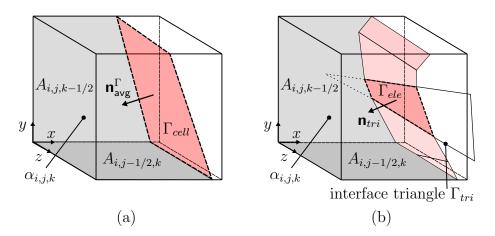


Fig. 3: Computation of cut-cell properties based on a level-set field Φ (a) and on intersection with a provided surface triangulation (b). For a detailed description of the cut algorithm please refer to [40].

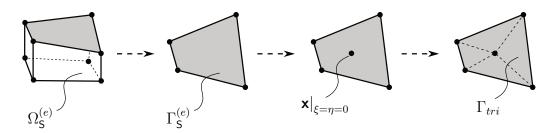


Fig. 4: Triangulation of an eight-node hexahedral element face (gray) contributing to the fluid-structure interface Γ .

et al. [40], these artifacts are mainly caused by a discontinuous evolution of fluid volume fractions when utilizing a level-set based representation of the interface. A solution to overcome these problems is to use an accurate representation of the geometry based on the computational fluid mesh and the provided structural interface. While the level-set method results in a planar approximation of the interface segment Γ_{cell} in a cell, see Fig. 3(a), the cut-element method recovers sub-cell interface resolution by a set of cutelements Γ_{ele} in a single fluid cell, see Fig. 3(b). The computation of the fluid volume fraction $\alpha_{i,j,k}$ is done by a sub-tetrahedralization of the fluid volume, while face apertures such as $A_{i,j-1/2,k}$ are calculated using a sub-triangulation of the cell faces [40].

A linear approximation of the possibly nonlinear structural interface is used for the cut algorithm as an input. The element surface $\Gamma_{\mathsf{S}}^{(e)}$ of an eightnode linear brick element, which contributes to the fluid-structure interface, is highlighted in gray, see Fig. 4. This surface is split into four interface triangles Γ_{tri} using an additional node at $\mathbf{x}|_{\xi=\eta=0}$ for improved approximation of its bilinear shape.

3.2.2. Interface exchange term

Interaction of the fluid with a solid interface is modeled by an interface exchange term $\chi_{i,j,k}$, as introduced in Eq. (20). Following the approach introduced above, we can write the interface exchange term as a sum of all individual contributions of all cut-elements contained within this computational cell,

$$\boldsymbol{\chi}_{i,j,k} = \sum_{ele} \boldsymbol{\chi}_{ele}.$$
(21)

For inviscid flows, the cut-element based interface exchange term χ_{ele} accounts for the pressure and pressure work at the element interface

$$\boldsymbol{\chi}_{ele} = \begin{bmatrix} 0 \\ \mathbf{p}_{ele}^{\Gamma} \Delta \Gamma_{ele} & \mathbf{n}_{1}^{\Gamma;ele} \\ \mathbf{p}_{ele}^{\Gamma} \Delta \Gamma_{ele} & \mathbf{n}_{2}^{\Gamma;ele} \\ \mathbf{p}_{ele}^{\Gamma} \Delta \Gamma_{ele} & \mathbf{n}_{3}^{\Gamma;ele} \\ \mathbf{p}_{ele}^{\Gamma} \Delta \Gamma_{ele} & (\mathbf{n}^{\Gamma;ele} \cdot \mathbf{u}^{\Gamma;ele}) \end{bmatrix}, \qquad (22)$$

where $\Delta\Gamma_{ele}$ is the element interface area, $\mathbf{n}^{\Gamma;ele} = [\mathbf{n}_1^{\Gamma;ele}, \mathbf{n}_2^{\Gamma;ele}, \mathbf{n}_3^{\Gamma;ele}]$ is the element unit normal vector obtained directly from the structural interface triangle Γ_{tri} , and $\mathbf{u}^{\Gamma;ele}$ is the interface velocity evaluated at the cut-element

face centroid. The element interface pressure $\mathsf{p}_{ele}^{\Gamma}$ is obtained by solving a symmetric face-normal Riemann problem

$$\mathcal{R}\left(\mathbf{w}_{i,j,k},\mathbf{u}^{\Gamma;ele}\right) = 0 \tag{23}$$

for each cut-element within the cut-cell (i, j, k). The exact solution of the reflective boundary Riemann problem (23) consists of either two shock waves $(\mathbf{u}_{i,j,k} \cdot \mathbf{n}^{\Gamma;ele} < \mathbf{u}^{\Gamma;ele} \cdot \mathbf{n}^{\Gamma;ele})$ or two rarefaction waves $(\mathbf{u}_{i,j,k} \cdot \mathbf{n}^{\Gamma;ele} \geq \mathbf{u}^{\Gamma;ele} \cdot \mathbf{n}^{\Gamma;ele})$, which are symmetric about the path of the moving interface coinciding with the contact wave [51]. The exact solution for the interface pressure $\mathbf{p}_{ele}^{\Gamma}$ is the root of

$$\left(\mathsf{p}_{ele}^{\Gamma}-\mathsf{p}_{i,j,k}\right)\cdot\sqrt{\frac{\frac{2}{(\gamma+1)\rho_{i,j,k}}}{\mathsf{p}_{ele}^{\Gamma}+\frac{\gamma-1}{\gamma+1}\mathsf{p}_{i,j,k}}}+\left(\mathbf{u}_{i,j,k}\cdot\mathbf{n}^{\Gamma;ele}-\mathbf{u}^{\Gamma;ele}\cdot\mathbf{n}^{\Gamma;ele}\right)=0\qquad(24)$$

for the two-shocks configuration, and

$$\mathbf{p}_{ele}^{\Gamma} = \mathbf{p}_{i,j,k} \cdot \left[1 + \left(\mathbf{u}^{\Gamma;ele} \cdot \mathbf{n}^{\Gamma;ele} - \mathbf{u}_{i,j,k} \cdot \mathbf{n}^{\Gamma;ele} \right) \cdot \frac{\gamma - 1}{2\sqrt{\gamma \mathbf{p}_{i,j,k}/\rho_{i,j,k}}} \right]^{\frac{2\gamma}{\gamma - 1}}$$
(25)

for the two-rarefactions configuration.

3.2.3. Boundary conditions for solid walls

Non-cut cells in the solid part of the computational domain in the vicinity of the interface contain ghost fluid states for imposing boundary conditions at the interface without requiring a modification of interpolation stencils in the finite volume reconstruction scheme. For this purpose, we apply the ghostcell methodology as proposed by Mittal et al. [37], extended to stationary and moving boundary cut-cell methods. Finding the ghost-cells and extending the fluid solution across the interface does not require the fully detailed cut-cell geometry. We perform this procedure based on the average face centroid and normal vector of the cut-cell, which is an average of all contained cut-elements weighted by their area. In a first step, ghost-cells \mathbf{x}_{GP} that contribute to the interpolation stencil of the baseline discretization are identified, see Fig. 5. Next, for each ghost-cell the boundary intercept point \mathbf{x}_{BI} is computed such that the line segment $\overline{\mathbf{x}_{GP}\mathbf{x}_{BI}}$ intersects the immersed boundary in \mathbf{x}_{BI} normal to the interface segment. The line segment is extended into the fluid region to find the image point

$$\mathbf{x}_{\mathsf{IP}} = \mathbf{x}_{\mathsf{BI}} + \mathbf{n}_{\mathsf{avg}}^{\Gamma} \cdot \Delta l, \qquad (26)$$

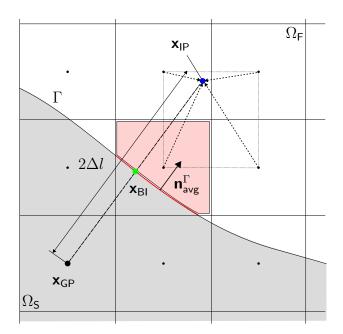


Fig. 5: Construction of the ghost-cell extending procedure for a cut-cell (i, j, k).

where $\Delta l = ||\mathbf{x}_{\mathsf{BI}} - \mathbf{x}_{\mathsf{GP}}||$ denotes the distance between the ghost-cell and the boundary intercept. Once the image point has been identified, a bilinear (in 2-D) or trilinear (in 3-D) interpolation is used for calculating the value of a quantity φ_{IP} at the image point \mathbf{x}_{IP} :

3D:
$$\varphi(x^{\star}, y^{\star}, z^{\star}) = c_1 + c_2 x^{\star} + c_3 y^{\star} + c_4 z^{\star} + c_5 x^{\star} y^{\star} + c_6 x^{\star} z^{\star} + c_7 y^{\star} z^{\star} + c_8 x^{\star} y^{\star} z^{\star}$$

2D: $\varphi(x^{\star}, y^{\star}) = c_1 + c_2 x^{\star} + c_3 y^{\star} + c_4 x^{\star} y^{\star},$ (27)

where $\mathbf{x}^{\star} = \mathbf{x} - \mathbf{x}_{\mathsf{IP}}$ is the relative distance vector and $\mathbf{c} = \{c_i\}$ are the unknown coefficients. As shown in Fig. 5, the four (eight in 3-D) coefficients can be determined from the variable values of the four (eight in 3-D) surrounding neighboring points,

$$\boldsymbol{c} = \boldsymbol{V}^{-1} \boldsymbol{\varphi} \,, \tag{28}$$

where φ denotes the solution at regular fluid data points and V^{-1} the inverse Vandermonde matrix, which is calculated by LU decomposition. After solving for (28), the value at the image point is given by

$$\varphi_{\mathsf{IP}} = c_1 + \mathcal{O}(\Delta^2) \,. \tag{29}$$

Ghost-cell values are obtained using a linear approximation along the line $\overline{\mathbf{x}_{GP}\mathbf{x}_{BI}}$ that satisfies the boundary conditions at the boundary intercept location \mathbf{x}_{BI} . For Dirichlet boundary conditions, ghost-cell data are obtained as

$$\varphi_{\mathsf{GP}} = 2 \cdot \varphi_{\mathsf{BI}} - \varphi_{\mathsf{IP}} + \mathcal{O}(\Delta l^2) \,, \tag{30}$$

whereas Neumann boundary conditions are imposed as

$$\varphi_{\mathsf{GP}} = \varphi_{\mathsf{IP}} - 2 \cdot \Delta l \left(\nabla \varphi \cdot \mathbf{n}_{\mathsf{avg}}^{\Gamma} \right) \Big|_{\mathbf{x}_{\mathsf{BI}}} + \mathcal{O}(\Delta l^2) \,. \tag{31}$$

The 5th-order WENO scheme used in this paper requires at least three layers of ghost-cells to be filled. This, in turn, poses a limitation of the current framework to structures with a size larger than several fluid cells in order to fill the ghost-cell values properly. An adaptive mesh refinement procedure for the flow solver or the decoupling of the ghost-cell method from the underlying Cartesian grid could resolve this limitation.

3.2.4. Treatment of small cut-cells

The time step Δt is adjusted dynamically according to the CFL condition based on full cells of the underlying Cartesian grid. A drawback of cutcell methods is that the fluid volume fraction of cut-cells may become very small and therefore can lead to numerical instability or require excessively small time steps with explicit time integration schemes and poor convergence with implicit methods. A stabilization of the underlying scheme is therefore required. We employ a so-called mixing procedure as proposed in [30, 40].

4. Numerical approach: Solid

The FEM is applied to solve the structural problem. Hence, we start with the weak form of the structural field equation, which is obtained by building weighted residuals of the balance equation (4) and Neumann boundary conditions (11) with virtual displacements δd . Subsequently, the divergence theorem is applied, yielding

$$\int_{\Omega_{\mathsf{S}}} \rho_{\mathsf{S};0} \, \ddot{\boldsymbol{d}} \cdot \delta \boldsymbol{d} \, \mathrm{d}V_0 + \int_{\Omega_{\mathsf{S}}} \boldsymbol{S} : \delta \boldsymbol{E} \, \mathrm{d}V_0 - \\ - \int_{\Omega_{\mathsf{S}}} \hat{\boldsymbol{b}}_0 \cdot \delta \boldsymbol{d} \, \mathrm{d}V_0 - \int_{\Gamma_{\mathsf{N};\mathsf{S}}} \hat{\boldsymbol{t}}_0 \cdot \delta \boldsymbol{d} \, \mathrm{d}A_0 - \delta \mathcal{W}_{\mathsf{S}}^{\Gamma} = \boldsymbol{0}$$
(32)

with infinitesimal volume and surface elements, dV_0 and dA_0 , respectively. Herein, $\delta \boldsymbol{E}$ is obtained as result of the variation of the Green-Lagrange strain (6), i.e.

$$\delta \boldsymbol{E} = \frac{1}{2} \left(\left(\nabla_0 \, \delta \boldsymbol{d} \right)^{\mathsf{T}} \cdot \boldsymbol{F} + \boldsymbol{F}^{\mathsf{T}} \cdot \nabla_0 \, \delta \boldsymbol{d} \right) \tag{33}$$

with $\nabla_0(\bullet)$ representing the material gradient operator. The influence of the interface on the structure is introduced via the additional virtual work term δW_5^{Γ} .

The weak form of equation (32) is discretized in space with the FEM. The solid domain Ω_{S} is split into n^{e} elements $\Omega_{\mathsf{S}}^{(e)}$ (subdomains). The semidiscrete weak form of the balance of linear momentum is obtained by assembling the contributions of all elements, leading to

$$\mathbf{M}\ddot{\mathbf{d}} + \mathbf{f}_{\mathsf{S};\mathsf{int}}(\mathbf{d}) - \mathbf{f}_{\mathsf{S};\mathsf{ext}}(\mathbf{d}) - \mathbf{f}_{\mathsf{S}}^{\Gamma} = \mathbf{0}, \qquad (34)$$

where we have assumed the discrete virtual displacement vector $\delta \mathbf{d}$ to be arbitrary. The vectors $\mathbf{\ddot{d}}$ and \mathbf{d} describe the discrete acceleration and displacement vectors, respectively, \mathbf{M} denotes the mass matrix, $\mathbf{f}_{S;int}$ and $\mathbf{f}_{S;ext}$ the internal and external force vectors. The interface traction of the fluid on the structure is described by \mathbf{f}_{S}^{Γ} . Element technology such as the method of enhanced assumed strains (EAS), as introduced in [49], is used in order to avoid locking phenomena. For time integration, the generalized trapezoidal rule (or one-step- θ scheme) is employed for the structure solver in this work. Thus, applying this scheme to the semi-discrete equation (34), the final fully discrete structural equation at the new time step n + 1 is obtained.

The fully discrete structural equation describes a system of nonlinear algebraic equations which is solved iteratively by a Newton-Raphson method. The linearized system reads

$$\mathbf{K}_{\mathsf{SS}}(\mathbf{d}_i^{n+1}) \,\Delta \mathbf{d}_{i+1}^{n+1} = -\mathbf{r}_{\mathsf{S}}(\mathbf{d}_i^{n+1}) \tag{35}$$

with iteration step *i*, the dynamic effective structural stiffness matrix \mathbf{K}_{SS} , and the residual vector \mathbf{r}_{S} . Thus, a new solution of the displacement increment $\Delta \mathbf{d}_{i+1}^{n+1}$ for current iteration step i + 1 is determined, and the final displacement solution of time step n + 1 is obtained via updating

$$\mathbf{d}_{i+1}^{n+1} = \mathbf{d}_{i}^{n+1} + \Delta \mathbf{d}_{i+1}^{n+1}.$$
(36)

The Newton-Raphson iteration is considered as converged if $|\mathbf{r}_{\mathsf{S}}|_2 \leq \epsilon$ is satisfied using a sufficiently small tolerance ϵ .

5. Coupling procedure

5.1. Treatment of non-matching interfaces

The reconstruction of the interface on the fluid side based on the structural position leads to a change in the number of cut-elements in each coupling step and to a change in connectivity, which inevitably results in a non-matching interface. A Mortar method has been chosen in this work as it preserves linear and angular momentum. The Mortar method requires the choice of a so-called slave and master side of the interface Γ^{sl} and Γ^{ma} , respectively. Primary coupling variables, such as velocities in our case, are transferred from the master to the slave side, and secondary variables, such as tractions, are transferred vice versa. The Dirichlet-Neumann partitioning chosen here determines the fluid to be the slave side ($\Gamma^{sl} \equiv \Gamma_F$) and the solid to be the master side ($\Gamma^{ma} \equiv \Gamma_S$) with respect to Mortar coupling. The aim is to obtain discrete projection operators for consistent data transferring.

In the following derivation, a no-slip condition between fluid and solid is assumed instead of the slip condition in (14) for simplicity, which will later be released again. The starting point is the weak form of the continuity constraint

$$\delta W_{\lambda} = \int_{\Gamma^{\rm sl}} \delta \boldsymbol{\lambda}^{\mathsf{T}} \left(\boldsymbol{u}^{\mathsf{\Gamma}} - \dot{\boldsymbol{d}}^{\mathsf{\Gamma}} \right) \mathrm{d}\Gamma = 0 \tag{37}$$

together with weak form of the equilibrium of tractions at the interface

$$\delta W_{\Gamma} = \int_{\Gamma^{\rm sl}} \boldsymbol{\lambda}^{\mathsf{T}} \left(\delta \boldsymbol{u}^{\mathsf{\Gamma}} - \delta \dot{\boldsymbol{d}}^{\mathsf{\Gamma}} \right) \mathrm{d}\Gamma$$
(38)

in which a Lagrange multiplier field $\lambda = \sigma_{\mathsf{F}}^{\Gamma} \cdot n^{\Gamma}$ and the corresponding test functions $\delta \lambda$ are introduced. The virtual work term (38) is the conjugate term of (37) and it contains virtual work contributions of interface tractions on the fluid side and on the solid side, $\delta \mathcal{W}_{F}^{\Gamma}$ and $\delta \mathcal{W}_{S}^{\Gamma}$, respectively. Additionally, $\delta \mathcal{W}_{S}^{\Gamma}$ needs to be adapted to the chosen time integration scheme for the solid due to the occurrence of the time derivative of the displacements.

An important question is which ansatz functions should be used for a proper interpolation of the respective fields at the interface. Due to the applied cut procedure in the underlying finite volume discretization it is not possible to obtain the surface ansatz functions for the cut-elements based on a trace space relationship. Without invoking high-order reconstruction, the FVM defines for the state values in the cut-cells a piecewise constant field as it is depicted in Fig. 6(a). For the solid, it is possible to obtain the ansatz

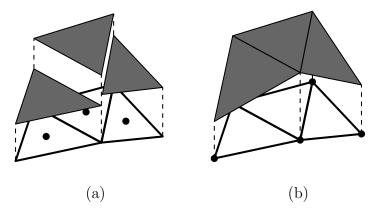


Fig. 6: Interpolation of state variables. (a) FVM: constant value per cell, (b) FEM: linear Lagrange polynomials

functions from the trace space of the underlying volume element leading to an interpolation with standard Lagrange polynomials as it is shown in Fig. 6(b). Hence, a Lagrange multiplier

$$\boldsymbol{\lambda} = \sum_{j=1}^{n^{\mathsf{sl}}} \Phi_j \lambda_j \tag{39}$$

using constant ansatz functions Φ_j on each cut-element can be utilized, which is defined only on the slave side of the interface. The discrete Lagrange multipliers are denoted as λ_j . Due to the constant value in a single cutelement also the velocities can be represented using constant ansatz functions $N_k^{\rm sl}$. This approach then reads

$$\mathbf{u} = \sum_{k=1}^{n^{\mathsf{sl}}} N_k^{\mathsf{sl}} \mathbf{u}_k.$$
(40)

In (39) and (40), the total number of cut-elements is denoted with n^{sl} , which is equal to the number of discrete fluid velocities \mathbf{u}_k due to the piecewise constant field on each cut-element. In contrast, standard shape functions N_l^{ma} based on Lagrange polynomials are used for the interpolation of the velocities on the solid side of the interface. This leads to

$$\dot{\mathbf{d}} = \sum_{l=1}^{n^{\text{ma}}} N_l^{\text{ma}} \dot{\mathbf{d}}_l \tag{41}$$

where the total number of discrete solid velocities \mathbf{d}_l is denoted as n^{ma} , which is equal to the number of nodes in the solid interface. Inserting (39) - (41) into (37) leads to

$$\delta W_{\lambda} = \sum_{j=1}^{n^{\mathsf{sl}}} \sum_{k=1}^{n^{\mathsf{sl}}} \delta \boldsymbol{\lambda}_{j}^{T} \left(\int_{\Gamma^{\mathsf{sl}}} \Phi_{j} N_{k}^{\mathsf{sl}} \,\mathrm{d}\Gamma \right) \mathbf{u}_{k} - \sum_{j=1}^{n^{\mathsf{sl}}} \sum_{l=1}^{n^{\mathsf{sl}}} \delta \boldsymbol{\lambda}_{j}^{T} \left(\int_{\Gamma^{\mathsf{sl}}} \Phi_{j} N_{l}^{\mathsf{ma}} \,\mathrm{d}\Gamma \right) \dot{\mathbf{d}}_{l} = 0.$$

$$(42)$$

Therein, nodal blocks of the two Mortar integral matrices commonly denoted as D and M can be identified. This leads to the following definitions:

$$\mathbf{D}[j,k] = D_{jk}\mathbf{I}_3 = \int_{\Gamma^{sl}} \Phi_j N_k^{sl} \,\mathrm{d}\Gamma\mathbf{I}_3\,,\tag{43}$$

$$\mathbf{M}[j,l] = M_{jl}\mathbf{I}_3 = \int_{\Gamma^{\mathsf{sl}}} \Phi_j N_l^{\mathsf{ma}} \,\mathrm{d}\Gamma\mathbf{I}_3 \tag{44}$$

with the 3×3 identity tensor I_3 , whose size is determined by the number of variables to be coupled for each node. Here, **D** is a square $3 n^{sl} \times 3 n^{sl}$ matrix, which has only diagonal entries due to the choice of piecewise constant shape functions, whereas the definition of **M** generally gives a rectangular matrix of dimensions $3 n^{sl} \times 3 n^{ma}$. The actual numerical integration of the Mortar integrals can be performed either segment-based or element-based, see [15, 19, 44, 45]. Due to its superior numerical efficiency, element-based integration is used exclusively in this work.

Plugging the previously defined Mortar matrices D and M into (37) leads to the discrete continuity constraint

$$\mathbf{D} \cdot \mathbf{u} - \mathbf{M} \cdot \dot{\mathbf{d}} = \mathbf{0},\tag{45}$$

which will be utilized in Section 5.1.1 for the specific transfer of velocities from the solid to the fluid interface. Similarly, inserting (39) - (41) into (38) and again using (43) and (44) results in

$$\mathbf{f}_{\mathsf{F}}^{\Gamma} = \mathbf{D}^{\mathsf{T}} \boldsymbol{\lambda} \,, \tag{46}$$

$$\mathbf{f}_{\mathsf{S}}^{\Gamma} = \mathbf{M}^{\mathsf{T}} \boldsymbol{\lambda} \,, \tag{47}$$

which defines the nodal coupling forces $\mathbf{f}_{\mathsf{F}}^{\Gamma}$ and $\mathbf{f}_{\mathsf{S}}^{\Gamma}$ of the fluid and the solid, respectively. The transfer of loads is based on (46) and (47) and will be described in Section 5.1.2.

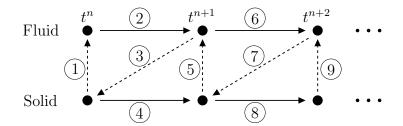


Fig. 7: Schematic of the staggered time integration of the coupled system.

5.1.1. Transfer of solid velocities to fluid interface

The velocity at the cut-element face centroid is needed for both the energy equation and for determining the interface pressure $\mathbf{p}_{ele}^{\Gamma}$ through a Riemann solver, see (22). Moreover, the kinematic constraint (14) requires matching normal velocities at the interface. In a first step, the full interface velocities are transferred to the fluid by reordering (45) and defining a discrete projection **P** operator, viz.

$$\mathbf{u} = \mathbf{D}^{-1} \cdot \mathbf{M} \, \dot{\mathbf{d}} = \mathbf{P} \, \dot{\mathbf{d}}. \tag{48}$$

It shall be noted that the inversion of \mathbf{D} is a trivial task at negligible cost due to its diagonal shape and thus there is no need for solving a possibly large linear system. In a second step, the current normal direction of the cut-element is used to project the velocity to fulfill the slip condition.

5.1.2. Transfer of fluid forces to solid interface

The equilibrium of forces requires the surface tractions of fluid and solid to be equal. As we do not want to solve explicitly for the Lagrange multipliers we reorder (46) and (47), yielding

$$\mathbf{f}_{\mathsf{S}}^{\Gamma} = \left(\mathbf{D}^{-1} \cdot \mathbf{M}\right)^{\mathsf{T}} \mathbf{f}_{\mathsf{F}}^{\Gamma} = \mathbf{P}^{\mathsf{T}} \mathbf{f}_{\mathsf{F}}^{\Gamma} .$$
(49)

One can see that the transfer of loads from the fluid to the solid is based on the transpose of the projection operator for the transfer of solid velocities to the fluid. This is a crucial requirement for the consistent transfer across the interface and a distinctive feature of Mortar methods.

5.2. Loosely coupled partitioned FSI algorithm

In this paper, we use a loosely coupled conventional serial staggered algorithm. In Fig. 7, we illustrate the main steps to advance the coupled system from time level t^n to $t^{n+1} = t^n + \Delta t^n$. This explicit staggering algorithm, which follows the classical Dirichlet-Neumann partitioning, reads as follows:

- 1. The known structural interface displacements $\mathbf{d}^{\Gamma;n}$ and velocities $\dot{\mathbf{d}}^{\Gamma;n}$ at time t^n are used to update the cut-cells list and geometric properties on the fluid side. For this purpose, the cut-element algorithm is applied on the triangulated structural interface (see Fig. 4).
- 2. Advance the fluid in time. The evaluation of the interface exchange term (22) and the prescription of ghost-cell values (30) and (31) at time t^{n+1} use given structural interface velocities $\dot{\mathbf{d}}^{\Gamma;n}$. An interpolation procedure is needed to transfer solid velocities to the fluid interface, see Section 5.1.1.
- 3. Transfer the fluid interface normal tractions $\boldsymbol{\sigma}_{\mathsf{F}}^{\Gamma;n+1} \cdot \mathbf{n}^{\Gamma;n}$ due to pressure loads to the structural solver. The staggering procedure leads to a time shift between the stress tensor and the normal used to compute the tractions. An interpolation procedure is needed to transfer fluid forces to the solid interface, see Section 5.1.2.
- 4. Advance the structure in time while the fluid interface loads act as additional Neumann boundary condition on the solid.
- 5. Proceed to the next time step.

Using the structural displacement $\mathbf{d}^{\Gamma;n}$ for the fluid solution at time t^{n+1} results in a first-order in time, $\mathcal{O}(\Delta t)$, coupling scheme [18]. Moreover, the explicit staggering algorithm is only conditionally stable since at time level t^{n+1} , the continuity condition is satisfied only for the dynamic part $(\boldsymbol{\sigma}_{\mathsf{F}}^{\Gamma;n+1} \cdot \mathbf{n}^{\Gamma;n})$ matches $\boldsymbol{\sigma}_{\mathsf{S}}^{\Gamma;n+1} \cdot \mathbf{n}^{\Gamma;n}$). For the kinematic part, the fluid velocities $\mathbf{u}^{\Gamma;n+1}$ at t^{n+1} match the structural velocities $\dot{\mathbf{d}}^{\Gamma;n}$ from the previous time step, but not the current structural velocities $\dot{\mathbf{d}}^{\Gamma;n+1}$. This in turn explains the violation of energy conservation at the interface.

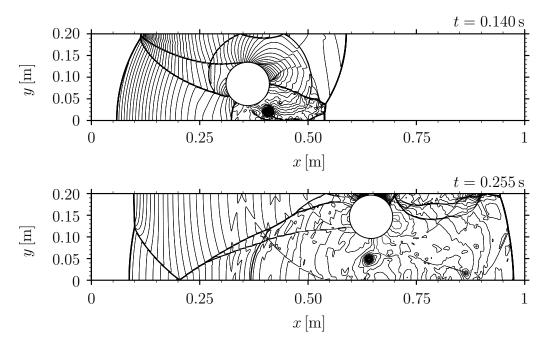


Fig. 8: 60 contours of fluid pressure within 0-28 Pa at two different time instances. Mesh resolution: $\Delta x = \Delta y = 6.25 \times 10^{-4}$ m

6. Validation of the FSI algorithm

In the following, we present a validation of our method for rigid and deformable structures. The solution of both subdomains $(\Omega_{\mathsf{F}}, \Omega_{\mathsf{S}})$ is advanced by the same time step which is based on the CFL condition for the fluid flow. For all examples, coupling is performed at every time step.

6.1. Shock wave impact on rigid cylinder

The following test case for rigid body motion has been originally proposed by Falcovitz et al. [14] and has been widely adopted in the literature, see e.g. [30, 39]. The setup consists of a two-dimensional channel filled with air and a rigid light-weight cylinder of density $\rho_{S,0} = 7.6 \text{ kg/m}^3$ initially resting on the lower wall at a position (x, y) = (0.15, 0.05) m. The cylinder is subsequently driven and lifted upwards by a Ma = 3 shock wave entering the domain from the left. The pre-shock conditions $\rho_{F;R} = 1 \text{ kg/m}^3$, $p_R =$ 1 Pa, $u_R = 0 \text{ m/s}$ hold for $x \ge 0.08 \text{ m}$ while for x < 0.08 m post-shock conditions $\rho_{F;L} = 3.857 \text{ kg/m}^3$, $p_L = 10.33 \text{ Pa}$, $u_L = 2.629 \text{ m/s}$ are initially

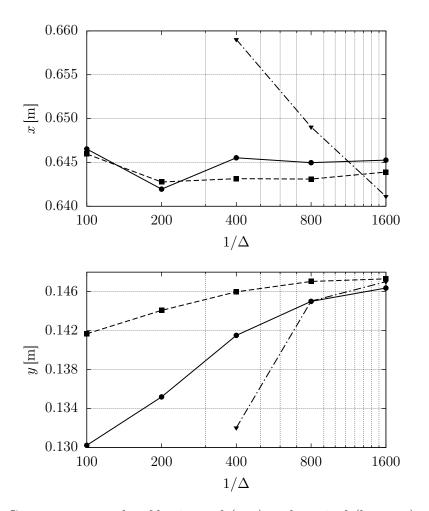


Fig. 9: Convergence study of horizontal (top) and vertical (bottom) cylinder center position for different fluid mesh resolutions. $(- \bullet -)$ present results, $(- \cdot - \nabla - \cdot -)$ Hu et al. [30], $(--\bullet -)$ Monasse et al. [39]. The *x*-axis is given in logarithmic scale.

prescribed. The fluid domain is rectangular with dimensions $1 \text{ m} \times 0.2 \text{ m}$ and is discretized with 1600×320 cells in streamwise and wall-normal direction, respectively. This leads to a grid resolution of $\Delta x = \Delta y = 6.25 \times 10^{-4}$ m. For the lower and upper wall, reflecting slip-wall boundary conditions are used. At the inflow the post-shock values are prescribed while a linear extrapolation of all flow variables is used at the outflow. The cylinder has a radius of $r = 0.05 \,\mathrm{m}$ and it is discretized with 240 tri-linearly interpolated hexahedral elements along its circumference, leading to 240 surface elements that are coupled to the fluid. Due to stability reasons the cylinder does not exactly rest on the lower wall initially. We found that a narrow gap equal to 2% of the local cell height leads to stable and accurate results. Rigidity is achieved by imposing a high Young's modulus. The time integration factor $\theta = 0.66$ is chosen for the structural time integration. A CFL number of 0.6 is adopted for all simulations. It should be noted that no analytical solution for the final position of the cylinder exists. We therefore put emphasis on convergence properties of the proposed coupling algorithm.

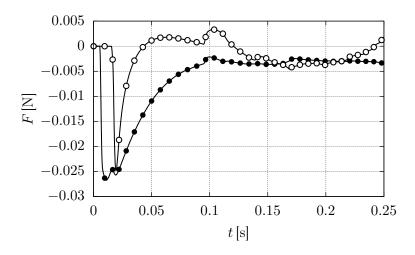


Fig. 10: Temporal evolution of global forces acting on the rigid cylinder. (----) F_x , (----) F_y . Mesh resolution: $\Delta x = \Delta y = 6.25 \times 10^{-4} \text{ m.}$

Instantaneous pressure contours at t = 0.14 s and t = 0.255 s are shown in Fig. 8. With respect to the cylinder position and the resulting shock patterns our results agree well to Fig. 19 of Hu et al. [30] and Fig. 11 of Monasse et al. [39]. We observe a strong vortex beneath the cylinder, which persists throughout the entire cylinder trajectory, see Fig. 8, supporting the results

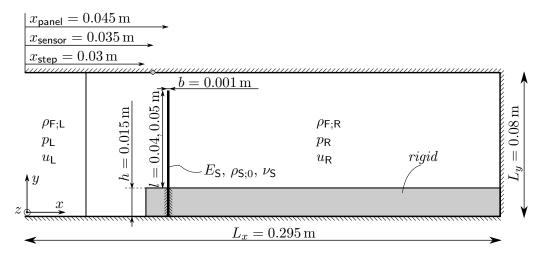


Fig. 11: Setup for shock wave impact on deforming panel including geometric dimensions; see [24] for details.

of [20, 39]. By further increasing the mesh resolution up to $\Delta x = \Delta y = 1.5625 \times 10^{-4}$ m the vortex is still apparent, excluding numerical dissipation being responsible for the formation of the vortex. As stated by Monasse et al. [39], a Kelvin-Helmholtz instability of the contact discontinuity present under the cylinder is the likely cause for this vortex.

Fig. 9 shows convergence results on the final horizontal and vertical position of the center of mass of the cylinder together with results from literature [30, 39]. The final position is in the same range as the results of [30, 39]. Our results show a convergence rate similar to the results obtained by Monasse et al. [39].

Finally, Fig. 10 shows the temporal evolution of resulting fluid forces acting on the rigid cylinder, which has been obtained by summation of all individual cut-element interface exchange terms. The smooth force distribution confirms that our interface treatment is accurate and free of spurious pressure oscillations.

6.2. Shock wave impact on deforming panel

The behavior of a cantilever panel subjected to a shock tube flow is analyzed. This test case has been investigated both experimentally and numerically in [24]. The experimental setup, as shown in Fig. 11, consists of a deformable panel of length l = [0.04, 0.05] m and width b = 0.001 m placed within a shock tube. The panel is hit by a Ma = 1.21 shock wave, which

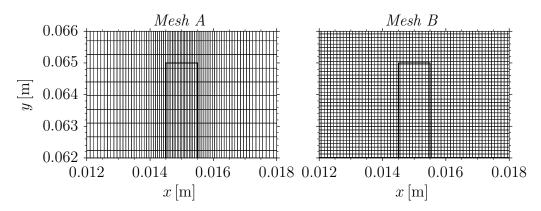


Fig. 12: Fluid mesh resolutions close to the panel.

enters the domain from left. The panel is made of steel ($E_{S} = 220 \text{ GPa}$, $\rho_{S:0} = 7600 \text{ kg/m}^3$, $\nu_S = 0.33$) and is clamped to a rigid forward-facing step at its lower end. The pre-shock conditions resemble air at rest and are set to $\rho_{\mathsf{F};\mathsf{R}} = 1.189 \,\mathrm{kg/m^3}, \ p_{\mathsf{R}} = 100 \,\mathrm{kPa}, \ u_{\mathsf{R}} = 0 \,\mathrm{m/s}, \ \mathrm{while \ the \ post-shock \ values}$ are $\rho_{\rm F;L} = 1.616 \, {\rm kg/m}^3$, $p_{\rm L} = 154 \, {\rm kPa}$, $u_{\rm L} = 109.68 \, {\rm m/s}$. The fluid domain is rectangular with dimensions $0.295 \,\mathrm{m} \times 0.08 \,\mathrm{m}$ in width and height. Since the problem is considered as two-dimensional, we adopt a constant thickness of 0.001 m in spanwise direction. Slip-wall boundary conditions are employed for all boundaries except for the inflow, where we prescribe non-reflective inflow boundary conditions based on Riemann invariants [43]. Two different fluid mesh resolutions are used: Mesh A contains 123,400 cells with grid stretching applied in flow direction close to the panel and Mesh B utilizes a homogeneous grid with 1.82 million cells, see Fig. 12. The panel is discretized using 65×2 (l = 0.05 m) or 55×2 (l = 0.04 m) tri-linearly interpolated hexahedral elements. For both cases the panel is fully clamped at the bottom, and symmetry boundary conditions are applied in spanwise direction. EAS is used in order to avoid shear locking, which may affect the solution in such bending-dominated problems when using first-order displacement-based elements. The time integration factor $\theta = 0.66$ is chosen for the structural time integration. A CFL number of 0.6 is set for all simulations.

Fig. 13 shows the time evolution of the horizontal displacement at the panel tip for the 0.05 m panel length case on *Mesh A* and *Mesh B*. The panel motion is almost identical for both fluid meshes throughout the entire simulation time. Results presented below are obtained on fluid *Mesh B*.

We start with a qualitative analysis of the flow field for the 0.05 m panel.

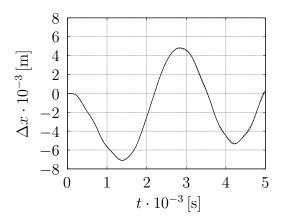


Fig. 13: Time evolution of panel tip displacement for 50 mm panel length using different fluid meshes. (-) Mesh A, (--) Mesh B

Fig. 14 shows numerical schlieren (left) and experimental shadowgraph visualizations (right) extracted from [24] at a time interval of $\Delta t = 140 \,\mu s$ for a time period of $T = 840 \,\mu s$. At $t = 0 \,\mu s$, the incident right-running shock wave has already hit the panel and base plate, leading to the formation of reflected and transmitted shock waves. Downstream of the panel the initially normal shock undergoes transition to a cylindrical shock front due to sudden area increase $(t = 140 \,\mu s)$. While being reflected at the lower wall $(t = 280 \,\mu s)$ and traveling downstream, it undergoes a transition from regular to Mach reflection $(t = 280 - 420 \,\mu s)$ and is subsequently reflected at the end wall $(t = 700 - 840 \,\mu s)$. A main vortex is initially produced at the panel tip due to the roll-up of the slipstream accompanied by a vortex shedding process. All flow characteristics described above match the experimental results without any notable time lag. However, three-dimensional effects due to leaks between the panel and the shock tube side walls are observed in the experiment $(t = 280 \,\mu s)$. Fig. 15 shows a numerical schlieren image at $t = 4.17 \,\mathrm{ms}$, illustrating the maximum panel deflection together with the interaction of the main vortex and the upstream moving shock wave.

A quantitative analysis is presented in Fig. 16, where the time evolution of the horizontal panel tip displacement is plotted. Fig. 16(a) refers to the 0.05 m panel length case and Fig. 16(b) to the 0.04 m case, respectively. In addition to experimental values [24] represented through error bars, we include recent inviscid numerical results of Sanches and Coda [48], who employed a finite element based partitioned FSI approach utilizing the ALE description

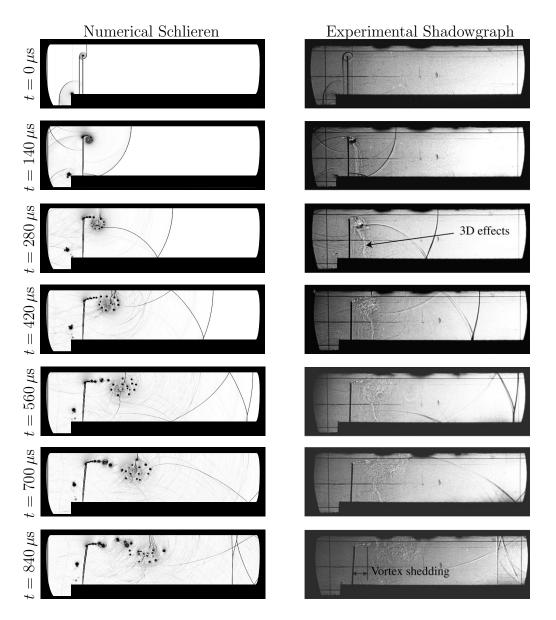


Fig. 14: Qualitative comparison between simulation (left) and experiment [24] (right) for 50 mm panel length by means of schlieren images for selected time instances.

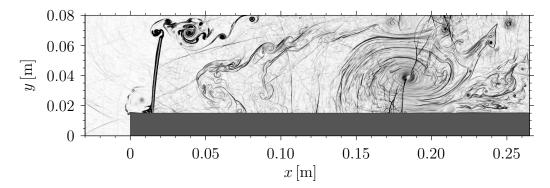


Fig. 15: Contour of density gradient magnitude at t = 4.17 ms.

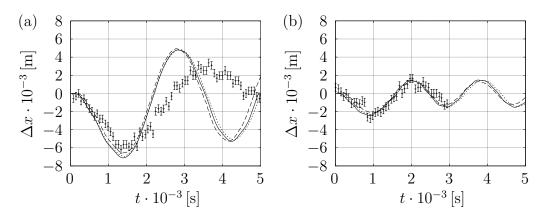


Fig. 16: Time evolution of panel tip displacement for (a) 0.05 m and (b) 0.04 m panel length. (—) present results, (--) Giordano et al. [24], (\cdots) Sanches and Coda [48]. Error bars denote experimental data [24].

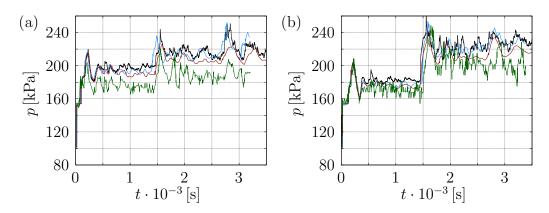


Fig. 17: Pressure signal recorded at sensor position (see Fig. 11 for exact location of pressure probe) for (a) 0.05 m and (b) 0.04 m panel length. (—) present results, (—) Giordano et al. [24], (—) Sanches and Coda [48], (—) experimental values [24]. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)

to account for moving boundaries and coupling with Lagrangian shell elements. Moreover, numerical results by Giordano et al. [24] are added, who assumed a two-dimensional but viscous flow in the laminar regime. For the 0.05 m panel case, see Fig. 16(a), it is observed that all numerical simulations predict a very similar oscillation of the panel with respect to the maximum amplitude and frequency of the first period. In comparison to the experimental values, both frequency and amplitude of the panel oscillation differ from numerical findings. According to Giordano et al. [24] this difference may be attributed to the lack of damping in the structural model, which, however, should be negligible at least for the first period. Another explanation given by the authors relates to small deformations of the base in the direct vicinity of the fixing point, which would slightly alter both frequency and amplitude of the panel motion. The panel oscillation period obtained with our method is 2.85 ms, which is very close to the analytical period of 2.87 ms when considering the first eigenmode of a clamped plate submitted to an impulse load [24]. The experimental period is given as 3.8 ms.

Due to these uncertainties, a second case with 0.04 m panel length has been studied experimentally and numerically in [24]. With the shorter panel, the stresses on the base part are reduced, which also diminishes the influence of the base on the panel motion. We observe excellent agreement with experimental data and numerical references, see Fig. 16(b).

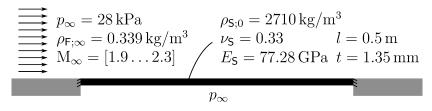


Fig. 18: Schematic and main parameters of the flutter problem.

Finally, the pressure signals recorded at (x, y) = (0.035, 0.08) m for both panel lengths are compared to the same numerical and experimental database in Fig. 17. Again, all numerical results are similar with respect to the time of arrival of pressure waves at the sensor and the pressure difference across the waves. While larger deviations are observed between numerical and experimental data for the 50 mm panel case, almost identical time evolution up to t = 2 ms is observed for the 0.04 m panel case. After that time, the pressure obtained experimentally drops continuously due to the arrival of reflected expansion waves inside the shock tube, which are not taken into account in the numerical simulations.

6.3. Flutter of a flat plate

Panel flutter is a self-excited, dynamic aeroelastic instability of thin plate structures, which frequently occurs in supersonic flow and is caused by an interaction between aerodynamic, inertial and elastic forces of the system [11]. For the setup considered here, see Fig. 18, linear instability theory predicts a critical Mach number of $\widetilde{Ma}_{crit} = 2.0$ above which a continuous growth of oscillations amplitudes is expected [12]. To trigger the instability, the pressure acting on the bottom of the panel initially is decreased by 0.1 % and is kept at this condition for 4 ms. After this time period, the pressure is set back to the free-stream pressure. Since the limit Mach number of $\widetilde{Ma}_{crit} =$ 2.0 describes a perfect oscillation without damping or amplification [42], this test case assesses effects of numerical damping present in our algorithm.

We consider a supersonic inviscid flow over a flat plate that is clamped at both ends, see Fig. 18. The plate has a length of l = 0.5 m, a thickness of t = 0.00135 m, a Young's modulus of $E_{\rm S} = 77.28$ GPa, a Poisson's ratio of $\nu_{\rm S} = 0.33$, and a density of $\rho_{\rm S;0} = 2710 \text{ kg/m}^3$. The structure is discretized using 200 × 8 tri-linearly interpolated hexahedral elements in streamwise and wall-normal direction, respectively. To avoid shear locking phenomena,

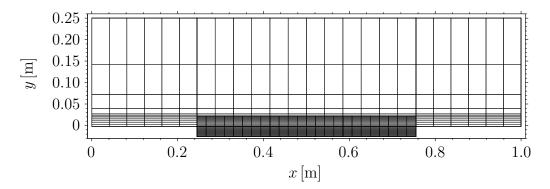


Fig. 19: General view of the computational domain and mesh resolution. Every 5th grid line is shown in the x- and y-direction, respectively.

the EAS method is used. Results obtained with tri-quadratically interpolated hexahedral elements and the same mesh resolution showed only negligible differences. If not stated otherwise, a geometric linear analysis of the structure is performed for comparison with references from the literature. The time integration factor $\theta = 0.5$ is chosen in order to reduce numerical damping. The fluid free-stream properties are: $\rho_{F,\infty} = 0.339 \,\text{kg/m}^3$, $p_{\infty} = 28 \,\mathrm{kPa}$ and Ma = [1.9...2.3]. The computational domain and the fluid mesh resolution is shown in Fig. 19. For the results presented here, a grid-converged solution with respect to the fluid domain has been obtained with a total number of 16,500 cells. The grid is uniform in the region around the panel (0.25 m $\leq x \leq$ 0.75 m) with a cell size of $\Delta x = 4.25 \times 10^{-3}$ m and $\Delta y = 4.8 \times 10^{-4}$ m. A cavity of height $h = 2.2 \times 10^{-2}$ m is added below the panel $(y \leq 0 \text{ m})$ to account for the panel motion in this region. Since the problem is two-dimensional, we adopted a constant thickness of $\Delta z = 5 \times 10^{-3}$ m in spanwise direction. Slip-wall boundary conditions are imposed at all boundary patches except for the inflow and outflow patch. At the inflow we prescribe all flow quantities which leads to a fully reflective boundary condition. At the outflow we perform linear extrapolation. The CFL number is 0.6 for all simulations.

The time evolution of the vertical displacement of the panel at the streamwise position x = 0.6 m for Mach numbers Ma = [1.9, 2.0, 2.05, 2.09, 2.1] is shown in Fig. 20(a). The gray shaded area indicates the initial perturbation time. While the panel oscillations for Mach numbers below Ma = 2.09 are damped, amplification of panel deflection can be observed for Ma = 2.1. We found the limit Mach number to be Ma_{crit} = 2.09, which is close to the

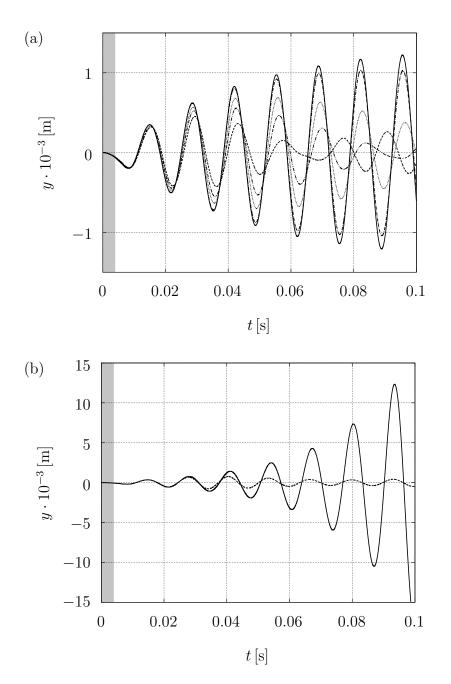


Fig. 20: (a) Vertical deflection of the plate at x = 0.6 m for Ma $\in [1.9, 2.0, 2.05, 2.09, 2.1]$. (- -) Ma = 1.9, (-..-) Ma = 2.0, (...) Ma = 2.05, (-..-) Ma = 2.09, (-..) Ma = 2.1. (b) Geometrically linear and nonlinear plate deflections at x = 0.6 m for Ma = 2.3. (-..) linear, (- -) nonlinear. The gray shaded area indicates the initial perturbation time.

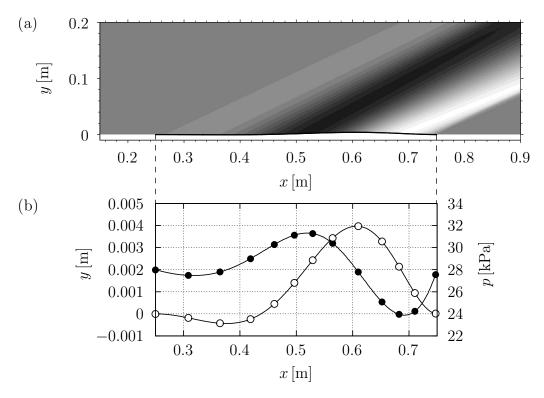


Fig. 21: (a) Pressure distribution for Ma = 2.3 at t = 0.068 s. Color scale from white to black using 20 equally spaced contour levels for $p \in [24 - 32]$ kPa. (b) Interface pressure and associated flutter mode. $(- \bullet -) p^{\Gamma}$, $(- \circ -) \Delta y$.

analytical solution ($\widetilde{Ma}_{crit} = 2.0$) with an error of 4.5% and to numerical results reported by Teixeira and Awruch [50] and Sanches and Coda [48] ($Ma_{crit} = 2.05$). Fig. 20(b) shows a comparison between geometrically linear and nonlinear panel solutions for a Mach number of Ma = 2.3. Exponential growth of the initial disturbance is observed for linear theory, which confirms analytical and numerical results [12, 42, 48, 50]. In the geometrically nonlinear case, limited displacement amplitudes are observed. According to Dowell [11], the behavior of the panel after flutter onset is mainly dominated by structural nonlinearities. Nonlinear structural coupling between bending and stretching of the plate may in fact increase its effective stiffness, thereby modifying the dynamic response of the system.

Finally, the pressure distribution in the fluid domain together with the

associated flutter mode and wall-pressure distribution at time instant t = 0.068 s is shown in Fig. 21. The deflection of the panel leads to the formation of compression and expansion waves in the fluid. Compression waves are observed for a positive interface slope, whereas expansion waves occur for negative interface slopes, which is consistent with Ackeret's linear theory. The maximum displacement for the flutter mode is found at 70% of the panel length, confirming analytical [12, 29] and numerical [42, 48, 50] findings. Local minima and maxima in the wall-pressure distribution in Fig. 21(b) coincide with interface inflection points. The smooth wall-pressure distribution confirms once again the accurate interface treatment.

6.4. Grid convergence study

The accuracy of the computed solution is verified through a grid convergence study. The simulation setup is similar to the case presented in Section 6.1. The formerly rigid cylinder is now replaced by an elastic structure and the wind-tunnel walls are removed. For the cylinder, which is initially located at (x, y) = (0.15, 0.0) m, a Young's modulus of $E_{\rm S} = 800$ Pa, a Poisson's ratio of $\nu_{\rm S} = 0.3$, and a density of $\rho_{\rm S;0} = 15$ kg/m³ have been adopted. The remaining parameters are identical to the setup described in Section 6.1.

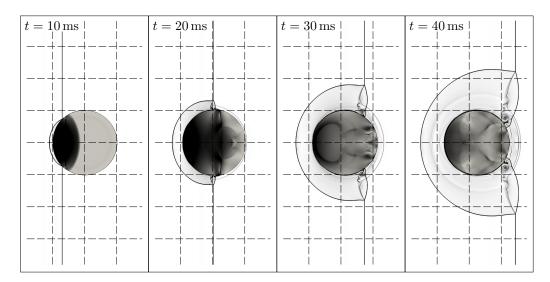


Fig. 22: Contours of density gradient magnitude in the fluid domain and magnitude of the Cauchy stress tensor in the solid domain at four different simulation times for the reference grid \mathcal{G}^{ref} .

Since no analytical solution for this complex interaction exists, we have performed a well resolved reference simulation. The reference grid, in the following denoted as \mathcal{G}^{ref} , has a resolution of 1280×2560 cells in the fluid domain and spatial dimensions of $0.2 \text{ m} \times 0.4 \text{ m}$. The cylinder is discretized with 2048 tri-linearly interpolated hexahedral elements along its circumference. For the remaining grids $\mathcal{G}^k|_{k=1...5}$, where \mathcal{G}^5 denotes the finest grid, the fluid resolution is successively halved and the unstructured mesh resolution of the solid is halved in radial and circumferential direction. A uniform time step of $\Delta t = 5.1 \times 10^{-6}$ s is used for all simulations, which corresponds to the maximum allowable time step size for the reference simulation at a CFL number of 0.6.

Figure 22 shows a numerical schlieren visualization of the resulting flow field together with the Cauchy stress field within the solid at times t =10, 20, 30 and 40 ms computed on \mathcal{G}^{ref} . As expected, both fields are symmetric with respect to the x axis, even though no symmetry is presumed for the algorithm. At time $t = 10 \,\mathrm{ms}$, the incident shock has already hit the cylinder and is subsequently reflected. The impact on the cylinder generates a shock wave which propagates through the solid. As the shock travels further around the cylinder, it undergoes transition from regular to Mach reflection (t = 20 ms). At the same time, the windward side of the cylinder is compressed, while the leeward side moves slightly downstream and generates a shock wave in the fluid. At the triple point, which connects the incident shock, the reflected shock and the Mach stem, a contact discontinuity develops. By the time the cylinder is accelerated (t = 30 ms), the reflected shock has propagated further upstream and a roll-up of the contact discontinuity is observed, which is enhanced by the interaction with the leeward shock wave. At the final time $t = 40 \,\mathrm{ms}$, several shock waves emerging from the fluid-structure interface can be observed and an overall complex flow field has developed.

As a qualitative measure of the accuracy of our method, Fig. 23 shows numerical schlieren visualizations together with Cauchy stresses at the final time t = 40 ms for all mesh resolutions \mathcal{G}^k . While the overall results with respect to the final cylinder position and the incident and reflected shock wave within the fluid domain agree well between all mesh resolutions, the finer grids ($\mathcal{G}^3, \mathcal{G}^4, \mathcal{G}^5$) provide fine scale features such as contact discontinuities and weak shock waves emerging from the cylinder surface which are partially missing or not well resolved on the coarse grids ($\mathcal{G}^1, \mathcal{G}^2$).

A quantitative measure of accuracy within the fluid domain is given by

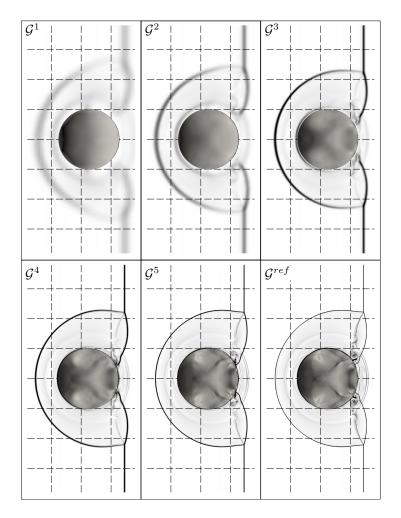


Fig. 23: Contours of density gradient magnitude in the fluid and magnitude of the Cauchy stress tensor in the solid domain at time t = 40 ms for all considered mesh resolutions.

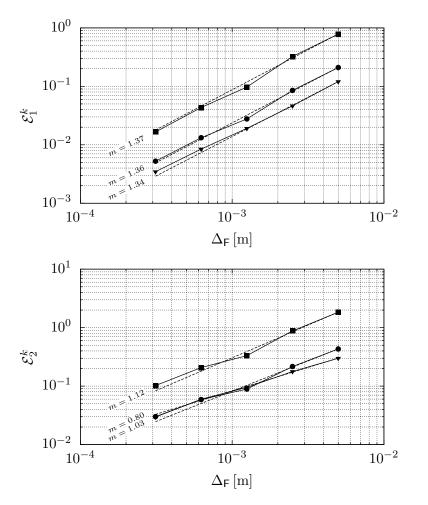


Fig. 24: Computed L_1 (top) and L_2 (bottom) fluid error norms \mathcal{E}_1^k and \mathcal{E}_2^k on all grids \mathcal{G}^k at time $t = 40 \text{ ms.} (- \bullet -) \mathcal{E}_{\rho}^k$, $(- \bullet -) \mathcal{E}_{p}^k$, $(- \bullet -) \mathcal{E}_{|\mathbf{u}|}^k$. Dashed lines represent least squares fits. Estimated convergence rates m are highlighted.

Table 1: Computed L_1 and L_2 fluid error norms \mathcal{E}_1^k and \mathcal{E}_2^k on all grids \mathcal{G}^k with respect to density, pressure and velocity magnitude at time t = 40 ms. Estimated convergence rates m are based on a least squares fit.

\mathcal{G}^k	$\Delta_{F}{}^{a}$	Δ_{S}^{b}	$\mathcal{E}^k_{1, ho}$	$\mathcal{E}^k_{2, ho}$	$\mathcal{E}^k_{1,p}$	$\mathcal{E}^k_{2,p}$	${\mathcal E}^k_{1, {f u} }$	$\mathcal{E}^k_{2, \mathbf{u} }$
$\begin{array}{c} \mathcal{G}^1 \\ \mathcal{G}^2 \\ \mathcal{G}^3 \\ \mathcal{G}^4 \\ \mathcal{G}^5 \end{array}$	$5 \cdot 10^{-3} \\ 2.5 \cdot 10^{-3} \\ 1.25 \cdot 10^{-3} \\ 6.25 \cdot 10^{-4} \\ 3.125 \cdot 10^{-4}$		$\begin{array}{c} 2.1 \cdot 10^{-1} \\ 8.5 \cdot 10^{-2} \\ 2.7 \cdot 10^{-2} \\ 1.3 \cdot 10^{-2} \\ 5.2 \cdot 10^{-3} \end{array}$	$\begin{array}{c} 4.3 \cdot 10^{-1} \\ 2.2 \cdot 10^{-1} \\ 8.9 \cdot 10^{-2} \\ 5.9 \cdot 10^{-2} \\ 3.0 \cdot 10^{-2} \end{array}$	$9.7 \cdot 10^{-2}$ $4.3 \cdot 10^{-2}$	$8.8 \cdot 10^{-1}$ $3.3 \cdot 10^{-1}$	$1.9 \cdot 10^{-2}$ $8.4 \cdot 10^{-3}$	$\begin{array}{c} 3.0 \cdot 10^{-1} \\ 1.7 \cdot 10^{-1} \\ 9.5 \cdot 10^{-2} \\ 6.0 \cdot 10^{-2} \\ 3.0 \cdot 10^{-2} \end{array}$
		Rate m	1.36	1.03	1.37	1.12	1.34	0.80

a. Fluid cell size in [m]. A uniform grid is used. *b*. Structural element length along the cylinder circumference in [m].

Table 2: Computed L_1 and L_2 structural error norms \mathcal{E}_1^k and \mathcal{E}_2^k with respect to the interface displacement magnitude and \mathcal{E}^k with respect to the interface force in x-direction on all grids \mathcal{G}^k at time t = 40 ms. Estimated convergence rates m are based on a least squares fit.

\mathcal{G}^k	$\Delta_{F}{}^a$	$\Delta_{S}{}^{b}$	$\mathcal{E}^k_{1,\left \mathbf{d}^\Gamma ight }$	$\mathcal{E}^k_{2,\left \mathbf{d}^\Gamma ight }$	$\mathcal{E}^k_{ig \pmb{\sigma}^{\Gamma}_{F} \cdot \mathbf{n}^{\Gamma}_{F} ig }$
\mathcal{G}^1	$5\cdot 10^{-3}$	$4.91\cdot 10^{-3}$	$8.9\cdot 10^{-4}$	$7.7\cdot 10^{-4}$	$5.6\cdot 10^{-3}$
\mathcal{G}^2	$2.5\cdot10^{-3}$	$2.45\cdot10^{-3}$	$6.1\cdot10^{-4}$	$5.4\cdot10^{-4}$	$2.0\cdot10^{-3}$
\mathcal{G}^3	$1.25\cdot 10^{-3}$	$1.23\cdot 10^{-3}$			
\mathcal{G}^4	$6.25\cdot10^{-4}$	$6.14\cdot10^{-4}$	$2.6\cdot 10^{-5}$	$2.4 \cdot 10^{-5}$	$2.2\cdot 10^{-4}$
\mathcal{G}^5	$3.125\cdot10^{-4}$	$3.07\cdot 10^{-4}$	$1.8 \cdot 10^{-5}$	$1.6 \cdot 10^{-5}$	$4.0 \cdot 10^{-5}$
		Rate m	1.58	1.56	1.75

a. Fluid cell size in [m]. A uniform grid is used. b. Structural element length along the cylinder circumference in [m].

the discrete L_p norm of the error for a solution variable \mathcal{S}^k on grid \mathcal{G}^k , which we define as

$$\mathcal{E}_{p}^{k} = \left[\frac{1}{N}\sum_{i=1}^{N} \left(\mathcal{S}_{i}^{k} - \mathcal{S}_{i}^{ref}\right)^{p}\right]^{\frac{1}{p}}.$$
(50)

Here, N denotes the total number of fluid cells considered on grid \mathcal{G}^k . In order to evaluate the convergence of the coupling problem, interface quantities are used. The error of the magnitude of the interface displacement of the structure is measured by integrating the error over the coupling surface. Thus, the sum in (50) is replaced by an integration and division by the number of grid points is replaced by division by the area of the coupling

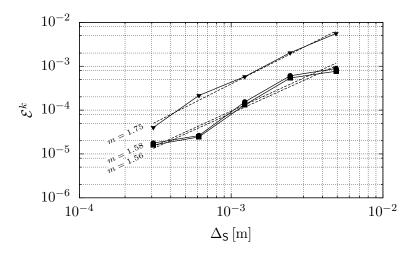


Fig. 25: Computed interface norms on all grids \mathcal{G}^k at time t = 40 ms. (— • —) $\mathcal{E}^k_{1,|\mathbf{d}^{\Gamma}|}$, (— • —) $\mathcal{E}^k_{2,|\mathbf{d}^{\Gamma}|}$, (— • —) $\mathcal{E}^k_{|\boldsymbol{\sigma}_{\mathsf{F}}^{\Gamma}\cdot\mathbf{n}_{\mathsf{F}}^{\Gamma}|}$. Dashed lines represent least squares fits. Estimated convergence rates m are highlighted.

surface. A second interface quantity of interest is the coupling force in x-direction. The corresponding error is computed as

$$\mathcal{E}^{k} = \left| \int_{\Gamma_{\mathsf{F}}^{k}} \boldsymbol{\sigma}_{\mathsf{F}}^{\Gamma,k} \cdot \mathbf{n}_{\mathsf{F}}^{\Gamma,k} \mathrm{d}\Gamma - \int_{\Gamma_{\mathsf{F}}^{ref}} \boldsymbol{\sigma}_{\mathsf{F}}^{\Gamma,ref} \cdot \mathbf{n}_{\mathsf{F}}^{\Gamma,ref} \mathrm{d}\Gamma \right| \,. \tag{51}$$

Table 1 summarizes estimated errors in the fluid density, pressure and velocity magnitude. Table 2 contains estimated errors in the structural interface displacement magnitude and in the coupling force in x-direction. Both tables include associated convergence rates at time t = 40 ms which are estimated from a least squares fit to the logarithm of the errors with the target function $\mathcal{F} = C \cdot \Delta^m$, where Δ denotes either the discrete fluid or structural mesh resolution and C denotes a positive constant independent of the grid. We observe convergence rates with respect to the L_1 norm of approximately 1.3 for all fluid variables, while the L_2 convergence rates are overall lower. Similar results have been observed by Henshaw and Schwendeman [28] for a pure fluid simulation of shock diffraction by a sphere. As expected from the flow field at time t = 40 ms, which is dominated by shock waves and contact discontinuities, the convergence order with respect to all fluid variables is first order. Figure 24 shows the associated variation of the L_1 and L_2 error norms in the fluid variables plotted against the fluid mesh resolution $\Delta x = \Delta y = \Delta_{\mathsf{F}}$. The error of the magnitude of the interface displacement is of order 1.58 and 1.56 in L_1 and L_2 norm, respectively. The convergence rate with respect to coupling force in x-direction computed at the interface from the fluid is 1.75. Due to the fluid-structure coupling and the use of tri-linear finite elements, the overall expected order of convergence is at most second order. Figure 25 contains the error norms for the magnitude of the interface displacements as well as for the coupling force in x-direction plotted against the structural mesh resolution Δ_{S} in circumferential direction.

This convergence study still has its limitations. First of all the almost standard limitation in such cases is not to appropriately take into account the coupling of spatial and temporal error but then comparing spatial errors at a certain point in time. An additional limitation in this case is that the specific FSI example does not include real structural dynamics in terms of large deformations, but rather shows a combination of rigid body dynamics combined with wave propagation in the solid, which has obviously different features. Given the lack of an established benchmark example we intended to stay close to a widely accepted example, namely the shock wave impact on a rigid cylinder as given before. Besides all the limitations, however, the provided convergence study should give some insight into the performance of the coupling approach.

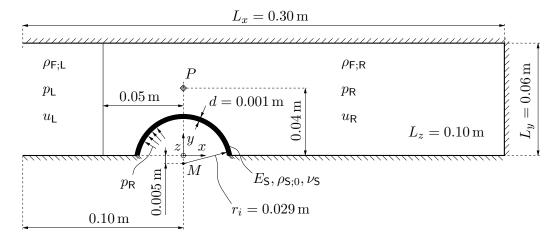


Fig. 26: Setup at xy-midplane for shock wave impact on a thin-walled shell including geometric dimensions.

7. Numerical Example - Buckling of a three-dimensional inflated thin shell

We present a numerical example to show the ability of our method to handle large and complex structural deformations in FSI problems. The presented example studies the interaction between a flexible inflated thin shell and a Ma = 1.21 shock wave. Pre- and post-shock fluid states are equal to the conditions introduced in Section 6.2, with the initial shock position being located at x = -0.05 m. Details of the setup are shown in Fig. 26. The spherical membrane has a thickness of d = 0.001 m and an inner radius $r_i =$ $0.029 \,\mathrm{m}$ with its center M located at $(x, y, z) = (0, -0.005, 0) \,\mathrm{m}$. Material properties are $E_{\rm S} = 0.07 \,{\rm GPa}, \ \rho_{\rm S;0} = 1000 \,{\rm kg/m}^3$ and $\nu_{\rm S} = 0.35$ for the Young's modulus, the density and the Poisson ratio, respectively. The thin shell is discretized with tri-linearly interpolated hexahedral elements with EAS, comprising two elements in thickness direction and 768 elements over the surface. The internal pressure keeping the membrane inflated is set equal to the pre-shock state $p_{\rm R}$. Zero displacements in all three directions are prescribed for structural nodes located at the bottom of the shell at y = 0 m. The time integration factor $\theta = 0.5$ is chosen.

Fig. 27 depicts the computational domain and the fluid mesh in xy- and xz-plane. In addition, we show the triangulated structural coupling interface, which is used for the cut process in the fluid solver. Slip-wall boundary conditions are applied to all boundaries except for the inflow, where all flow

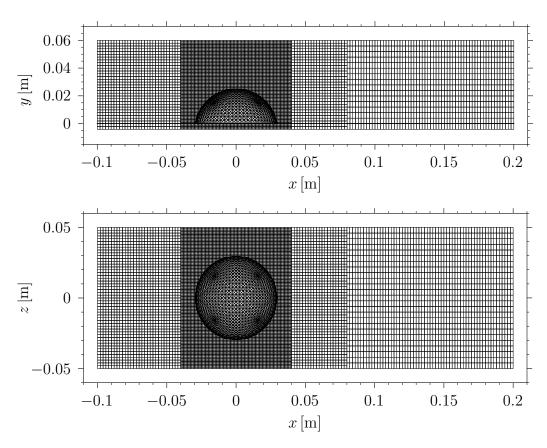


Fig. 27: General view of the computational domain and mesh resolution. The triangulated solid interface is additionally illustrated.

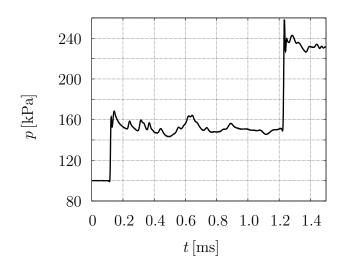
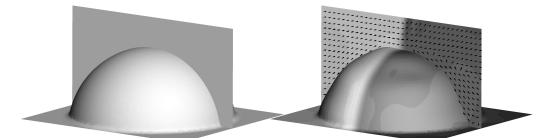


Fig. 28: Pressure signal recorded at sensor position (x, y, z) = (0, 0.04, 0) m.

quantities are prescribed leading to a fully reflective boundary condition. In the region around the shell, a uniform grid is used with cell sizes $\Delta x = \Delta y = \Delta z = 0.001$ m. In total, the fluid domain is discretized in space with 616,000 cells. The time step size is chosen to match a CFL number of 0.6.

In Fig. 28, the time evolution of the pressure signal recorded at the sensor position P with (x, y, z) = (0, 0.04, 0) m is shown. The jumps at approximately t = 0.1167 ms and at t = 1.2196 ms mark the times when the shock wave passes the sensor. Pressure distributions and velocity vectors at the xymidplane are shown in Fig. 29 for different time instances. The corresponding strain distribution in the thin-walled shell is presented in Fig. 30. The norm of the Euler-Almansi strain tensor $|e|_2 = \sqrt{e \cdot e}$ evaluated at each element center of the top layer is chosen to illustrate the large deformations occurring during the buckling process. Initially, the structure is undeformed and stress-free and the fluid is at rest, see Fig. 29(a) and Fig. 30(a). Due to the overpressure induced by the shock, Fig. 29(b), the windward side of the membrane is compressed, see Fig. 30(b), and is subsequently bouncing back due to its elastic behavior. At time $t = 0.7704 \,\mathrm{ms}$, buckling of the thin-walled shell occurs at its tip, deflecting the flow as depicted in Fig. 29(c). The displacement of the tip node at initial position (x, y, z) = (0, 0.025, 0) m (monitoring point A) is given in Fig. 31(a): the y-deflection is approximately 1.5×10^{-3} m during this first shock induced dimpling process. As the shock hits the membrane after reflection at the end wall, see Fig. 29(d)-(f), the pressure increases



(a) $t = 0 \text{ ms}, p \in [100 - 170] \text{ kPa}$ (d) $t = 1.22 \text{ ms}, p \in [107 - 260] \text{ kPa}$



(b) $t = 0.285 \,\mathrm{ms}, p \in [100 - 170] \,\mathrm{kPa}$ (e) $t = 1.4 \,\mathrm{ms}, p \in [107 - 260] \,\mathrm{kPa}$



(c) $t = 0.7704 \text{ ms}, p \in [100 - 170] \text{ kPa}$ (f) $t = 1.5 \text{ ms}, p \in [107 - 260] \text{ kPa}$

Fig. 29: Pressure distribution together with uniform-length velocity vectors at different time instances. Every second vector is shown on the xy-midplane. Color scale ranges from white to black using 20 equally spaced contour levels within the indicated pressure range.

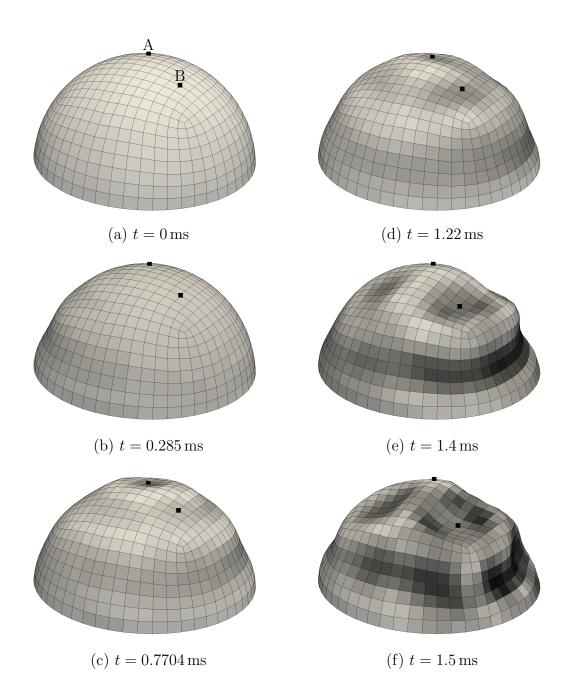


Fig. 30: Norm of the Euler-Almansi strain tensor evaluated at each element center over time. Color scale ranges from white to black using 26 equally spaced contour levels for $|\mathbf{e}|_2 \in [0 - 0.13]$. Monitoring points A and B are marked with squares.

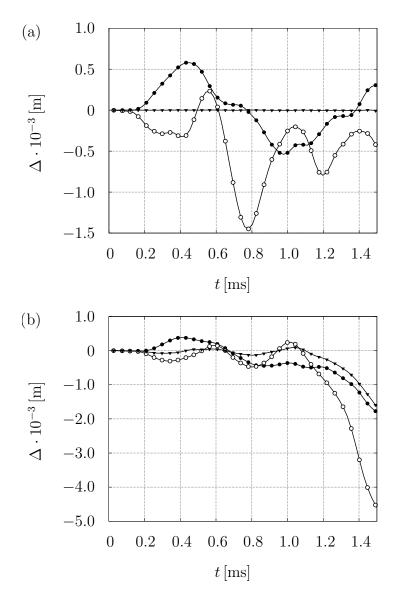


Fig. 31: Time evolution of displacements at two different monitoring points (see Fig. 30): (a) tip of membrane: monitoring point A, (b) monitoring point B. (— • —) Δx , (— • —) Δy , (— $\mathbf{\nabla}$ —) Δz

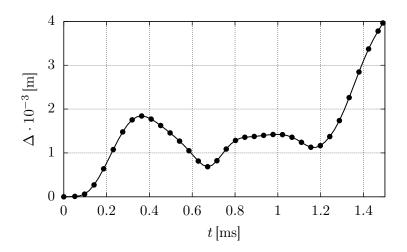


Fig. 32: Time evolution of integral displacement magnitude.

again. The membrane cannot sustain the additional load, and we observe the formation of buckling dimples, which are symmetrically distributed with respect to the xy-midplane as shown in Fig. 29(d)-(f) and Fig. 30(d)-(f). At t = 1.5 ms, the norm of the Euler-Almansi strain in the most distorted regions rises up to 0.127, see Fig. 30(f). Considering the monitoring point B, which is initially located at (x, y, z) = (0.011912, 0.020912, 0.009308) m in one of the dimples, a total deflection of 5.23×10^{-3} m is found, see Fig. 31(b).

We refined the grids for both subdomains simultaneously and separately (not shown here for brevity) in order to reveal sensitivities with respect to the dynamic response of the thin-walled membrane. While the displacement of the membrane does not change significantly when varying the fluid resolution and keeping the structural discretization the same (maximum relative error of 2% compared to a fluid grid with $\Delta x = \Delta y = \Delta z = 0.00025$ m), we found that the dynamic response of the membrane and especially the occurring buckling mode can depend on the structural resolution. This observation confirms that buckling is highly sensitive with respect to imperfections of all kinds, including geometric imperfections [46]. Reliable prediction of buckling modes require realistic imperfection models, derived from the particular manufacturing process, to be included in the numerical model, which is beyond the scope of this paper. A well-defined quantity for such a configuration is the integral displacement magnitude shown in Fig. 32. A maximum relative error of 3% is found when comparing the present results to those of a four times finer mesh for both subdomains. A grid converged solution with respect to the integral displacement is obtained for a twice finer mesh. Figure 32 shows that the membrane starts to collapse at around t = 1.2 ms, which coincides with the time when the shock wave, after reflection at the end wall, passes the pressure sensor P.

8. Summary and conclusions

The proposed finite volume – finite element coupling approach for the interaction between a compressible fluid and a deformable structure is able to handle large and complex three-dimensional deformations. We make use of a classical Dirichlet-Neumann partitioning in conjunction with a conventional serial staggered procedure for coupling of the two domains.

A representation of the interface within the fluid domain is achieved by means of a cut-element based IBM, which has been successfully extended to deformable structures for the first time. The presented framework leads to a non-matching discretization of the interface between both subdomains. A consistent data transfer has been established using a Mortar method, which preserves linear and angular momentum. Piecewise constant ansatz functions are used for interpolating the fluid state as well as for the Lagrange multipliers on each single cut-element, allowing for a simple inversion of a diagonal matrix at negligible cost for the evaluation of the discrete projection operator. To the authors knowledge, this is the first time a cut-element method has been combined with a Mortar method for coupling the two subdomains in a consistent and efficient way.

The proposed coupling method has been validated through two-dimensional model problems involving rigid and deformable structures with large deformations. Our method correctly predicts the transient behavior of shockloaded rigid and deformable structures. Moreover, good accuracy was achieved with respect to the correct prediction of flutter onset. The ability of our method to handle three-dimensional FSI problems involving large and complex structural deformations has been demonstrated through a newly proposed test case consisting of a flexible inflated thin shell interacting with a shock wave.

The current framework is limited to structures with a size larger than several fluid cells in order to fill the ghost-cell values properly. A remedy could be either an adaptive mesh refinement procedure for the flow solver or the decoupling of the ghost-cell method from the underlying Cartesian grid, which leads to additional degrees of freedom that need to be handled. In order to resolve the possibly different time-scales of both subdomains and increase the overall efficiency, subcycling should be considered for future work.

Acknowledgements

The authors gratefully acknowledge support by the German Research Foundation (Deutsche Forschungsgemeinschaft) in the framework of the Collaborative Research Centre SFB/TRR 40 "Fundamental Technologies for the Development of Future Space-Transport-System Components under High Thermal and Mechanical Loads". Computational resources have been provided by the Leibniz Supercomputing Centre of the Bavarian Academy of Sciences and Humanities (LRZ).

Appendix A. Computational Performance

Case (# Run)	$N_{\rm F}^{ele}$	$N_{\sf S}^{ele}$	$\frac{N_{F}^{ele}}{N_{F}^{CPU}}$	$\frac{N_{\sf S}^{ele}}{N_{\sf S}^{CPU}}$	$T_{F}(\%)$	$T_{S}\left(\% ight)$	$T_{C}\left(\% ight)$
$\mathbf{Cylinder}^{a}$							
#1	$2 \cdot 10^{3}$	$3.6\cdot 10^3$	$2 \cdot 10^3$	$0.9\cdot 10^3$	1.7%	98.1%	0.2%
#2	$8\cdot 10^3$	$3.6\cdot 10^3$	$8\cdot 10^3$	$0.9\cdot 10^3$	4.4%	95.4%	0.2%
#3	$3.2\cdot 10^4$	$3.6\cdot 10^3$	$8\cdot 10^3$	$0.9\cdot 10^3$	5.0%	94.8%	0.2%
#4	$1.28\cdot 10^5$	$3.6\cdot 10^3$	$1.6\cdot 10^4$	$0.9\cdot 10^3$	9.4%	90.4%	0.2%
#5	$5.12\cdot 10^5$	$3.6\cdot 10^3$	$6.4\cdot 10^4$	$0.9\cdot 10^3$	28.5%	71.3%	0.2%
\mathbf{Panel}^{b}							
#1	$-1.234 \cdot 10^{5}$	$1.3 \cdot 10^2$	$1.12\cdot 10^4$	$4.3\cdot 10^1$	56.0%	43.4%	0.6%
#2	$1.82\cdot 10^6$	$1.3\cdot 10^2$	$6.07\cdot 10^4$	$6.5\cdot 10^1$	45.9%	53.9%	0.2%
$\mathbf{Flutter}^{c}$							
#1	$1.65 \cdot 10^4$	$1.6\cdot 10^3$	$4.125\cdot 10^3$	$1.3\cdot 10^2$	13.9%	85.5%	0.6%
Membrane							
#1	$6.16 \cdot 10^{5}$	$1.536\cdot 10^3$	$3.08\cdot 10^4$	$1.28\cdot 10^2$	45.2%	54.2%	0.6%

Table A.3: Computational performance of the coupling framework for selected simulations.

a. Only the rigid cylinder case is considered. b. Only the 50 mm panel length case is considered. c. Only the Ma = 2.3 case is considered.

The performance of the proposed coupling algorithm is summarized in Table A.3, where we show the percentage of time spent for the fluid solver T_{F} ,

the structural solver T_{S} , and for the communication T_{C} between both codes for all considered test cases. $N_{\mathsf{F},\mathsf{S}}^{ele}$ represents the total number of elements used for the fluid and structural problem, respectively. $N_{\mathsf{F},\mathsf{S}}^{ele}/N_{\mathsf{F},\mathsf{S}}^{CPU}$ is the associated number of elements per CPU for each subdomain. The majority of the computational time is spent on advancing the solid domain, which, however, also includes load transfer with the Mortar method. Increasing the fluid resolution proportionally increases the number of cut-elements and thus the workload for the structural solver at the interface. Moreover, the implicit time integration leads to an iterative solution procedure with at least two Newton iterations per coupling step to obtain the solid state. The communication between both codes via Message Passing Interface typically requires less than 1% of the runtime. The current implementation of the staggered algorithm can be further optimized in terms of parallel efficiency. Furthermore, subcycling can significantly reduce computational cost of the structural solver and will be considered in future work.

References

References

- Banks, J., Henshaw, W., Schwendeman, D., 2012. Deforming composite grids for solving fluid structure problems. Journal of Computational Physics 231 (9), 3518–3547.
- [2] Banks, J., Henshaw, W., Schwendeman, D., 2014. An analysis of a new stable partitioned algorithm for FSI problems. Part I: Incompressible flow and elastic solids. Journal of Computational Physics 269, 108–137.
- [3] Banks, J., Henshaw, W., Sjögreen, B., 2013. A stable FSI algorithm for light rigid bodies in compressible flow. Journal of Computational Physics 245, 399–430.
- [4] Bernardi, C., Maday, Y., Patera, A. T., 1994. A new nonconforming approach to domain decomposition: the mortar element method. In: Brezis, H., Lions, J. (Eds.), Nonlinear partial differential equations and their applications. Pitman/Wiley: London/New York, pp. 13–51.
- [5] Causin, P., Gerbeau, J.-F., Nobile, F., 2005. Added-mass effect in the design of partitioned algorithms for fluid-structure problems. Computer Methods in Applied Mechanics and Engineering 194, 4506–4527.

- [6] Cirak, F., Deiterding, R., Mauch, S. P., 2007. Large-scale fluid-structure interaction simulation of viscoplastic and fracturing thin-shells subjected to shocks and detonations. Computers & Structures 85, 1–33.
- [7] Clarke, D. K., Hassan, H. A., Salas, M. D., 1986. Euler calculations for multielement airfoils using Cartesian grids. AIAA Journal 24, 353–358.
- [8] Colella, P., Graves, D. T., Keen, B. J., Modiano, D., 2006. A Cartesian grid embedded boundary method for hyperbolic conservation laws. Journal of Computational Physics 211, 347–366.
- [9] de Boer, A., van Zuijlen, A. H., Bijl, H., 2007. Review of coupling methods for non-matching meshes. Computer Methods in Applied Mechanics and Engineering 196, 1515–1525.
- [10] Donea, J., Giuliani, S., Halleux, J., 1982. An arbitrary lagrangianeulerian finite element method for transient dynamic fluid-structure interactions. Computer Methods in Applied Mechanics and Engineering 33, 689–723.
- [11] Dowell, E. H., 1972. NASA Space Vehicle Design Criteria. Monograph NASA SP-8005.
- [12] Dowell, E. H., 1974. Aeroelasticity of plates and shells. Springer, Leyden.
- [13] Ehrl, A., Popp, A., Gravemeier, V., Wall, W. A., 2014. A dual mortar approach for mesh tying within a variational multiscale method for incompressible flow. International Journal for Numerical Methods in Fluids 76, 1–27.
- [14] Falcovitz, J., Alfandary, G., Hanoch, G., 1997. A two-dimensional conservation laws scheme for compressible flows with moving boundaries. Journal of Computational Physics 138, 83–102.
- [15] Farah, P., Popp, A., Wall, W. A., 2015. Segment-based vs. element-based integration for mortar methods in computational contact mechanics. Computational Mechanics 55, 209–228.
- [16] Farhat, C., Lesoinne, M., 2000. 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, 499–515.

- [17] Farhat, C., Lesoinne, M., Le Tallec, P., 1998. Load and motion transfer algorithms for fluid/structure interaction problems with non-matching discrete interfaces: Momentum and energy conservation, optimal discretization and application to aeroelasticity. Computer Methods in Applied Mechanics and Engineering 157, 95–114.
- [18] Farhat, C., Lesoinne, M., Maman, N., 1995. Mixed explicit/implicit time integration of coupled aeroelastic problems: Three-field formulation, geometric conservation and distributed solution. International Journal for Numerical Methods in Fluids 21, 807–835.
- [19] Fischer, K. A., Wriggers, P., 2006. Mortar based frictional contact formulation for higher order interpolations using the moving friction cone. Computer Methods in Applied Mechanics and Engineering 195, 5020– 5036.
- [20] Forrer, H., Berger, M., 1999. Flow simulations on Cartesian grids involving complex moving geometries. Hyperbolic problems: theory, numerics, applications 129, 315–324.
- [21] Förster, C., Wall, W. A., Ramm, E., 2007. Artificial added mass instabilities in sequential staggered coupling of nonlinear structures and incompressible viscous flows. Computer Methods in Applied Mechanics and Engineering 196, 1278–1293.
- [22] Gaffney, R. L., Hassan, H. A., 1987. Euler calculations for wings using Cartesian grids. In: 25th AIAA Aerospace Sciences Meeting. American Institute of Aeronautics and Astronautics, Reston, Virginia.
- [23] Garelli, L., Paz, R. R., Storti, M. A., 2010. Fluid-structure interaction study of the start-up of a rocket engine nozzle. Computers & Fluids 39, 1208–1218.
- [24] Giordano, J., Jourdan, G., Burtschell, Y., Medale, M., Zeitoun, D. E., Houas, L., 2005. Shock wave impacts on deforming panel, an application of fluid-structure interaction. Shock Waves 14, 103–110.
- [25] Gottlieb, S., Shu, C.-W., 1998. Total variation diminishing Runge-Kutta schemes. Mathematics of Computation of the American Mathematical Society 67, 73–85.

- [26] Grilli, M., Hickel, S., Hu, X. Y., Adams, N. A., 2009. Conservative Immersed Interface Method for compressible viscous flows with heat transfer. In: Proceedings of the Academy Colloquium Immersed Boundary Methods: Current Status and Future Research Directions. KNAW, Amsterdam (The Netherlands).
- [27] Günther, C., Meinke, M., Schröder, W., 2014. A flexible level-set approach for tracking multiple interacting interfaces in embedded boundary methods. Computers & Fluids 102, 182–202.
- [28] Henshaw, W. D., Schwendeman, D. W., 2008. Parallel computation of three-dimensional flows using overlapping grids with adaptive mesh refinement. Journal of Computational Physics 227, 7469–7502.
- [29] Houbolt, J. C., 1958. A study of several aerothermoelastic problems of aircraft structures in high-speed flight. Ph.D. thesis, ETH Zürich.
- [30] Hu, X. Y., Khoo, B. C., Adams, N. A., Huang, F. L., 2006. A conservative interface method for compressible flows. Journal of Computational Physics 219, 553–578.
- [31] Joppich, W., Kürschner, M., 2006. MpCCI a tool for the simulation of coupled applications. Concurrency and Computation: Practice and Experience 18, 183–192.
- [32] Kirkpatrick, M. P., Armfield, S. W., Kent, J. H., 2003. A representation of curved boundaries for the solution of the Navier–Stokes equations on a staggered three-dimensional Cartesian grid. Journal of Computational Physics 184, 1–36.
- [33] Klöppel, T., Popp, A., Küttler, U., Wall, W. A., 2011. Fluid-structure interaction for non-conforming interfaces based on a dual mortar formulation. Computer Methods in Applied Mechanics and Engineering 200, 3111–3126.
- [34] Küttler, U., Wall, W. A., 2008. Fixed-point fluid-structure interaction solvers with dynamic relaxation. Computational Mechanics 43, 61–72.
- [35] Liu, X. D., Osher, S., Chan, T., 1994. Weighted essentially nonoscillatory schemes. Journal of Computational Physics 115, 200–212.

- [36] Meyer, M., Devesa, A., Hickel, S., Hu, X. Y., Adams, N. A., 2010. A conservative immersed interface method for Large-Eddy Simulation of incompressible flows. Journal of Computational Physics 229, 6300–6317.
- [37] Mittal, R., Dong, H., Bozkurttas, M., Najjar, F. M., Vargas, A., von Loebbecke, A., 2008. A versatile sharp interface immersed boundary method for incompressible flows with complex boundaries. Journal of Computational Physics 227, 4825–4852.
- [38] Mittal, R., Iaccarino, G., 2005. Immersed boundary methods. Annual review of fluid mechanics 37, 239–261.
- [39] Monasse, L., Daru, V., Mariotti, C., Piperno, S., Tenaud, C., 2012. A conservative coupling algorithm between a compressible flow and a rigid body using an Embedded Boundary method. Journal of Computational Physics 231, 2977–2994.
- [40] Orley, F., Pasquariello, V., Hickel, S., Adams, N. A., 2015. Cut-element based immersed boundary method for moving geometries in compressible liquid flows with cavitation. Journal of Computational Physics 283, 1–22.
- [41] Peskin, C. S., 1972. Flow patterns around heart valves: a numerical method. Journal of Computational Physics 10, 252–271.
- [42] Piperno, S., 1997. Explicit/implicit fluid/structure staggered procedures with a structural predictor and fluid subcycling for 2D inviscid aeroelastic simulations. International Journal for Numerical Methods in Fluids 25, 1207–1226.
- [43] Poinsot, T., Lele, S. K., 1992. Boundary Conditions for Direct Simulations Compressible Viscous Flows. Journal of Computational Physics 101, 104–129.
- [44] Popp, A., Gitterle, M., Gee, M. W., Wall, W. A., 2010. A dual mortar approach for 3D finite deformation contact with consistent linearization. International Journal for Numerical Methods in Engineering 83, 1428– 1465.
- [45] Puso, M. A., 2004. A 3D mortar method for solid mechanics. International Journal for Numerical Methods in Engineering 59, 315–336.

- [46] Ramm, E., Wall, W. A., 2004. Shell structures a sensitive interrelation between physics and numerics. International Journal for Numerical Methods in Engineering 60 (1).
- [47] Ross, M. R., Felippa, C. A., Park, K. C., Sprague, M. A., 2008. Treatment of acoustic fluidstructure interaction by localized Lagrange multipliers: Formulation. Computer Methods in Applied Mechanics and Engineering 197, 3057–3079.
- [48] Sanches, R. A. K., Coda, H. B., 2014. On fluid-shell coupling using an arbitrary Lagrangian-Eulerian fluid solver coupled to a positional Lagrangian shell solver. Applied Mathematical Modelling 55, 1–18.
- [49] Simo, J. C., Armero, F., 1992. Geometrically non-linear enhanced strain mixed methods and the method of incompatible modes. International Journal for Numerical Methods in Engineering 33, 1413–1449.
- [50] Teixeira, P. R. F., Awruch, A. M., 2005. Numerical simulation of fluidstructure interaction using the finite element method. Computers & Fluids 34, 249–273.
- [51] Toro, E. F., 2009. Riemann Solvers and Numerical Methods for Fluid Dynamics. Springer Berlin Heidelberg, Berlin, Heidelberg.
- [52] Udaykumar, H. S., Shyy, W., Rao, M. M., 1996. Elafint: a mixed Eulerian-Lagrangian method for fluid flows with complex and moving boundaries. International Journal for Numerical Methods in Fluids 22, 691–712.
- [53] Wohlmuth, B. I., 2000. A mortar finite element method using dual spaces for the Lagrange multiplier. SIAM Journal on Numerical Analysis 38, 989–1012.
- [54] Ye, T., Mittal, R., Udaykumar, H. S., Shyy, W., 1999. An accurate Cartesian grid method for viscous incompressible flows with complex immersed boundaries. Journal of Computational Physics 156, 209–240.
- [55] Zhao, X., Bayyuk, S., Zhang, S., 2013. Aeroelastic response of rocket nozzles to asymmetric thrust loading. Computers & Fluids 76, 128–148.