A combined Finite Volume – Finite Element approach for fluid-structure interactions using a cut-cell Immersed Boundary method

By V. Pasquariello†, F. Örley†, S. Hickel† and N. A. Adams†
G. Hammerl‡, C. Danowski‡, A. Popp‡ and W. A. Wall‡

† Institute of Aerodynamics and Fluid Mechanics, Technische Universität München
‡ Institute of Computational Mechanics, Technische Universität München
Boltzmannstr. 15, Garching bei München, Germany

We present a loosely coupled approach for the solution of three-dimensional fluid-structure interaction problems between a compressible fluid and a deformable structure. The method is based on a staggered Dirichlet-Neumann partitioning. In contrast to common algorithms employing the Arbitrary Lagrangian-Eulerian (ALE) description on body-fitted grids, the interface movement employed in the Eulerian flow solver is accounted for by a conservative sharp-interface Immersed Boundary (IB) method. The present method strictly enforces geometric conservation by directly computing parameters of the cut plane based on a triangulated fluid-structure interface. Special attention is paid to the correct transfer of loads between both sub-domains at the non-matching conjoined interface. While velocities of the solid are transferred to the fluid domain through a simple and efficient interpolation scheme, a projection algorithm inspired by the dual mortar method is used for the interpolation of fluid loads onto the solid interface. Numerical simulations for two- and three-dimensional inviscid compressible flows interacting with shock-loaded deformable structures are analyzed and results are validated against numerical and experimental references.

1. Introduction

The simulation of Fluid-Structure Interaction (FSI) problems is one of the most challenging disciplines in Computational Fluid Dynamics (CFD), covering many applications ranging from nonlinear aeroelasticity [1–3] to shock-induced deformations of rocket nozzles [4–6]. The long-term objective of the present investigation is to correctly predict the transient FSI of a fully turbulent boundary layer over an elastic plate subjected to an incident shock wave [7]. While focusing on this kind of application, we predominantly consider deformable structures subjected to shock loads.

FSI algorithms may generally be classified into monolithic and partitioned approaches. While monolithic methods treat the fluid and structure dynamics in the same mathematical framework to form a single system of equations for the entire simulation domain, partitioned algorithms handle the two domains as disjoint computational fields connected through suitable interface conditions [8]. One main advantage often attributed to monolithic systems is their superior numerical stability due to a unified discretization basis. This unified discretization, however, at the same time imposes limitations regarding FSI
applications. While a Lagrangian description is classically used in structural mechanics for the numerical simulation of large solid deformations, shock-capturing methods for compressible flows are usually formulated in an Eulerian frame of reference. As the numerical applications become more complex, highly advanced specialized solvers for both sub-domains are needed. Hence, partitioned algorithms for FSI are often used because of their great advantage to combine efficient and validated solvers for each subproblem. The idea of a temporal splitting technique goes back to the early work of [9, 10] and is often referred to as Staggered Solution Procedure. Partitioned methods can be further distinguished into loosely and strongly coupled algorithms, depending on whether the coupling conditions are exactly satisfied at each time step, or not. While partitioned algorithms can be made strong through the introduction of equilibrium iterations [4, 11], loosely coupled approaches are still frequently used in the field of aeroelasticity and compressible viscous fluids [2, 12–15]. However, a well-known disadvantage of weakly coupled partitioned algorithms is the added mass effect [14, 16–18], which may lead to numerical instability and is especially prevalent in incompressible flow descriptions and for high fluid-solid density ratios. For these kind of configurations the use of equilibrium iterations is indispensable. Another general classification of FSI methods is based upon 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 [19–22], and fictitious domain methods [23–28]. ALE approaches make use of body-fitted grids and thus need a mesh-movement step to account for time-varying solid interfaces. This is known to be a non-trivial and costly task, especially when computing problems that involve large solid displacements. Non-conforming fictitious domain methods, on the other hand, 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. Since in general no re-meshing process is needed, these methods are also capable to handle topological changes, such as crack propagation [13, 29]. A brief overview of fictitious domain methods, which in the literature are also often referred to as Immersed Boundary Methods (IBM) or Embedded Boundary Methods (EBM), is given in the following.

The computation of the flow field around complex and moving geometries on simple Cartesian grids with IBM goes back to the pioneering work of [23]. IBM can broadly be distinguished into continuous and discrete forcing techniques [30]. The continuous forcing technique introduces a source term into the governing equations, which is smeared over multiple cells across the fluid-solid interface to fulfill the boundary condition in an approximate manner [31–34]. Such techniques have been proven to handle geometries of large complexity including moving obstacles [35, 36]. However, a well-known drawback of these methods is the spurious loss or production of mass, momentum and energy at the interface [37, 38]. Such non-conservativity poses a particular issue for Large-Eddy Simulations (LES), which employ coarse grids and rely on an accurate prediction of physical flow behavior 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 [39] and [40] for inviscid flows and later extended to viscous flows by [41–43] and [44]. 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 [45]. 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 [44, 46], cell-linking [47] or flux redistribution techniques [48–51].

In this paper, we develop a loosely coupled approach for the solution of three-dimensional fluid-structure interaction problems between a compressible fluid and a deformable structure. The interface movement employed in the Eulerian flow solver is accounted for by a conservative sharp-interface Immersed Boundary method [52, 53]. While previous methods often reconstruct the solid boundary based on a level-set function, we resort to a more detailed representation of the fluid-solid interface by considering individual cut-elements within one single fluid cell for achieving sub-cell resolution of the interface. A positive side-effect is the absence of numerical artifacts, such as spurious pressure oscillations near the interface. Special attention is paid to the correct transfer of loads between both sub-domains at the non-matching conjoined interface. While solid velocities are transferred to the fluid domain using a simple and efficient interpolation scheme, a projection algorithm inspired by the dual mortar method is utilized for the interpolation of fluid loads to the solid interface.

This paper is structured as follows: First, the governing equations for fluid and solid together with the fluid-structure interface conditions are introduced in Section 2. Subsequently, Section 3 gives a detailed overview on the numerical treatment of moving boundaries within the Eulerian flow solver. Moreover, the discretization methods for the inviscid compressible fluid and the deformable structure are presented. In Section 4, the loose coupling approach is presented together with the treatment of non-matching interfaces. In Section 5, our method is validated against well-established two-dimensional test cases. Subsequently, in Section 5.4, we show results for the interaction between a flexible inflated membrane and a shock wave, illustrating in particular the ability of our FSI approach to handle large and complex three-dimensional deformations. Finally, a summary and concluding remarks are outlined in Section 6.

2. Mathematical and physical model

In the following, mathematical and physical models for both fluid (Section 2.1) and solid (Section 2.2) domain are outlined. Fluid-structure interface conditions are discussed in Section 2.3.

2.1. Governing equations for the fluid

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

$$\frac{\partial w}{\partial t} + \nabla \cdot K(w) = 0 \text{ in } \Omega_F.$$  (2.1)

The state vector \( w = [\rho_F, \rho_F u_1, \rho_F u_2, \rho_F u_3, E_t] \) contains the conserved variables density \( \rho_F \), momentum \( \rho_F u \) and total energy \( E_t \). The subscript \( F \) denotes fluid quantities and is used whenever a distinction between both sub-domains is necessary. The individual
contributions of the flux tensor $K = (f, g, h)$ are given as

$$f(w) = \begin{pmatrix} \rho_F u_1 \\ \rho_F u_1^2 + p \\ \rho_F u_1 u_2 \\ u_1(E_t + p) \end{pmatrix}, \quad g(w) = \begin{pmatrix} \rho_F u_2 \\ \rho_F u_2^2 + p \\ \rho_F u_2 u_3 \\ u_2(E_t + p) \end{pmatrix}, \quad h(w) = \begin{pmatrix} \rho_F u_3 \\ \rho_F u_3^2 + p \\ \rho_F u_3 u_4 \\ u_3(E_t + p) \end{pmatrix},$$

where $p$ is the static pressure. We consider a perfect gas with a constant Prandtl number of $Pr = 0.72$, specific-heat ratio of $\gamma = 1.4$ and specific gas constant of $R = 287.058 \frac{J}{kgK}$. Pressure $p$ and temperature $T$ are determined by the ideal-gas equation of state

$$p = \rho_F RT$$

and the definition of total energy

$$E_t = \frac{1}{\gamma - 1} p + \frac{1}{2} \rho_F u_i u_i.$$  

Equations including several index pairs are expanded successively by the Einstein summation convention.

### 2.2. Governing equations for the structural domain

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

$$\rho_S \ddot{d} = \nabla_0 \cdot (F \cdot S) + b_0 \text{ in } \Omega_S,$$

describing equilibrium of the forces of inertia, internal and external forces in the undeformed structural domain $\Omega_S$. 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 $\rho_S$. Furthermore, $d$ and $\ddot{d}$ are the unknown displacements and accelerations, respectively. The vector field $b_0$ is the material body force, where the superimposed hat identifies a prescribed quantity. 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 are applicable. For the sake of simplicity, in this work a hyperelastic Saint Venant-Kirchhoff material model with strain energy density function $\Psi$ per unit reference volume is chosen as

$$\Psi(E) = \mu_S E : E + \frac{1}{2} \lambda_S (E : I)^2,$$

where the Lamé constants $\lambda_S$ and $\mu_S$ and the second-order identity tensor $I$ are utilized. The Green-Lagrange strain tensor is defined as

$$E = \frac{1}{2} (F^T \cdot F - I).$$

The second Piola-Kirchhoff stress

$$S = \frac{\partial \Psi}{\partial E}$$

is derived using (2.6). An alternative stress measure represents the first Piola-Kirchhoff stress tensor $P$ defined as

$$P = F \cdot S.$$
A combined FV-FE approach for FSI using a cut-cell IBM

parts:

$$\partial \Omega_S = \Gamma_{S,D} \cup \Gamma_{S,N} \cup \Gamma.$$  

(2.10)

The common interface $\Gamma$ between the distinct domains of the fluid and solid, $\Omega_F$ and $\Omega_S$, respectively, will be discussed in detail in Section 2.3. On the Dirichlet boundary $\Gamma_{S,D}$, the displacements are prescribed, whereas on the Neumann boundary $\Gamma_{S,N}$, the traction vector $\mathbf{t}_0$ is prescribed using the unit normal vector $\mathbf{n}_0$ in material configuration. Thus, the boundary conditions

$$d = \hat{d} \quad \text{on } \Gamma_{S,D},$$  

(2.11)

$$P \cdot \mathbf{n}_0 = \hat{\mathbf{t}}_0 \quad \text{on } \Gamma_{S,N}.$$  

(2.12)

need to be satisfied.

The balance equation (2.5) describes a second-order partial differential equation in time, hence initial conditions for both displacements $d$ and velocities $\dot{d}$ need to be specified at time $t = 0$ following

$$d_0 = d(X, t = 0) = \hat{d}_0 \quad \text{on } \Omega_S,$$  

(2.13)

$$\dot{d}_0 = \dot{d}(X, t = 0) = \hat{\dot{d}}_0 \quad \text{on } \Omega_S,$$  

(2.14)

where $X$ defines the initial position.

2.3. Fluid-structure interface conditions

Dynamic and kinematic coupling conditions at the conjoined interface $\Gamma = \Omega_F \cap \Omega_S$ ensure the integrity between the sub-domains in this partitioned coupling algorithm. Assuming no mass to be transported across the interface, the normal velocities have to match, i.e.

$$\mathbf{u}^\Gamma \cdot \mathbf{n}^\Gamma = \frac{\partial d^\Gamma}{\partial t} \cdot \mathbf{n}^\Gamma \quad \text{on } \Gamma,$$  

(2.15)

where $\mathbf{n}^\Gamma$ denotes the interface unit normal vector. Equivalently, the dynamic condition requires the tractions to be equal, hence

$$\mathbf{\sigma}^\Gamma \cdot \mathbf{n}^\Gamma = \mathbf{P}^\Gamma \cdot \mathbf{n}_0^\Gamma \quad \text{on } \Gamma,$$  

(2.16)

where $\mathbf{\sigma}^F = p \mathbf{I}$ denotes the fluid stress tensor comprising only contributions due to the pressure in the inviscid case considered here. The normal vector $\mathbf{n}^\Gamma$ in the spatial configuration, as well as its counterpart $\mathbf{n}_0^\Gamma$ in the material configuration, point outward from the solid and into the fluid domain.

3. Numerical approach

3.1. Fluid: Finite Volume Immersed Interface Method

We use a flow solver based on the Finite Volume Method (FVM) for solving the Euler equations on Cartesian grids. The time-dependent fluid-solid interface conditions on $\Gamma$ are imposed through a conservative Immersed Interface method (CIIM).

3.2. Mathematical model

The computational domain $\Omega$ is divided into a fluid and solid domain, $\Omega_F$ and $\Omega_S$, which are separated by the interface $\Gamma$. In the following, $\Gamma$ accordingly denotes the fluid-structure interface of the continuous problem, and $\Gamma_{F/S}$ the flow and structure side of
the interface of the discrete problem. We solve the integral form of (2.1),
\[
\int_{t^n}^{t^{n+1}} \int_{\Omega_{i,j,k} \cap \Omega_F} \left( \frac{\partial w}{\partial t} + \nabla \cdot K(w) \right) dV dt = 0,
\]
where the integral is taken over the volume \( \Omega_{i,j,k} \cap \Omega_F \) of a computational cell \((i, j, k)\) and time step \( \Delta t = t^{n+1} - t^n \). Applying Gauss’ theorem results in
\[
\int_{t^n}^{t^{n+1}} \int_{\Omega_{i,j,k} \cap \Omega_F} \frac{\partial w}{\partial t} dV dt + \int_{t^n}^{t^{n+1}} \int_{\partial(\Omega_{i,j,k} \cap \Omega_F)} K(w) \cdot n dS dt = 0,
\]
where \( \partial(\Omega_{i,j,k} \cap \Omega_F) \) 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
\[
w_{i,j,k} = \frac{1}{\alpha_{i,j,k} V_{i,j,k}} \int_{\Omega_{i,j,k} \cap \Omega_F} w dV dS dZ,
\]
leads to
\[
\begin{align*}
\alpha_{i,j,k}^{n+1} w_{i,j,k}^{n+1} &= \alpha_{i,j,k}^n w_{i,j,k}^n + \frac{\Delta t}{\Delta x_i} \left[ A_{i-1/2,j,k}^n f_{i-1/2,j,k} - A_{i+1/2,j,k}^n f_{i+1/2,j,k} \right] \\
&+ \frac{\Delta t}{\Delta y_j} \left[ A_{i,j-1/2,k}^n g_{i,j-1/2,k} - A_{i,j+1/2,k}^n g_{i,j+1/2,k} \right] \\
&+ \frac{\Delta t}{\Delta z_k} \left[ A_{i,j,k-1/2}^n h_{i,j,k-1/2} - A_{i,j,k+1/2}^n h_{i,j,k+1/2} \right] \\
&+ \frac{\Delta t}{V_{i,j,k}} X_{i,j,k}.
\end{align*}
\]
\( V_{i,j,k} = \Delta x_i \Delta y_j \Delta z_k \) corresponds to the total volume of cell \( \Omega_{i,j,k} \), \( \alpha_{i,j,k} \) corresponds to the fluid volume fraction, \( w_{i,j,k} \) is the vector of volume-averaged conserved quantities in the cut-cell, and \( A \) is the effective fluid wetted cell face aperture. The face
A combined FV-FE approach for FSI using a cut-cell IBM

Figure 2. Computation of geometrical cut-cell properties based on a level-set field $\Phi$ (a) and on exact intersection with a provided surface triangulation (b) [53].

Average numerical fluxes over the cell faces are denoted as $f$, $g$, and $h$. Time integration of the state vector is shown here for a forward Euler time integration scheme with a timestep $\Delta t$, which, e.g., corresponds to one sub-step when using a higher-order Runge-Kutta method. For all simulations presented in this paper we employ a spatial flux discretization in characteristic space by an adaptive central-upwind-$5^{th}$-order accurate WENO scheme [54] together with a Lax-Friedrichs flux function. A $3^{rd}$-order total variation diminishing Runge-Kutta scheme [55] is used for the time integration. A sketch of a two-dimensional cut-cell is shown in Fig. 1. The flux $\chi_{i,j,k}$ across the interface $\Gamma_{i,j,k} = \Gamma \cap \Omega_{i,j,k}$, which is only present in cells cut by the interface, is discussed in detail below.

3.3. Conservative immersed boundary method

The fluid-structure interface $\Gamma$ is represented by a sharp-interface cut-cell method as described in the following section.

3.3.1. Geometry computation

With previous approaches [52, 56, 57] the geometric properties of the cut-cell (fluid face apertures $A$, interface unit normal vector $n$, and the fluid volume fraction $\alpha_{i,j,k}$) were obtained from a level-set field $\Phi$, which represents a signed distance of a point in the domain from the immersed boundary. As a result, a piecewise linear approximation of the interface $\Gamma$ is obtained, as shown in Fig. 2 (a). This method has been successfully applied for simple stationary boundaries, see [52, 56–58]. When dealing with moving boundaries featuring sharp corners and complex geometries, on the other hand, numerical artifacts in terms of spurious pressure oscillations can develop, which are primarily caused by the violation of the geometric conservation law (GCL) near the immersed boundary [59].

A solution to strictly enforce geometric conservation is to use a more accurate reconstruction of the geometrical parameters. Following the work of [53], we resort to the exact representation of the geometry based on the computational fluid mesh and the provided interface triangulation, as shown in Fig. 2(b). While the level-set method leads to an averaged interface $\Gamma_{cell}$ as an approximation of the geometry, this cut-element method represents the interface as a number of cut-elements $\Gamma_{ele}$. Each cut-cell contains and updates its respective individual set of cut-elements, which is also used to
compute the interface exchange term $\chi_{i,j,k}$ for each cut-cell as described below. The computation of the exact geometry is done by a sub-triangulation of the cell faces and a sub-tetrahedralization of the fluid volume [53]. Fig. 3 illustrates the triangulation of a structural interface element $\Gamma_S^{(e)}$, which serves as an input for the cut-algorithm. The element face of an eight-node linear brick element, which contributes to the fluid-structure interface, is highlighted in gray. An additional node at $x_{\xi=\eta=0}$ is added to the element face in order to account for its bilinear shape function. In this example, the triangulation yields four interface triangles $\Gamma_{tri}$ per structural interface element $\Gamma_S^{(e)}$.

3.3.2. Interface exchange term

Interaction of the fluid with a solid interface is modelled by an interface exchange term $\chi_{i,j,k}$, as introduced in (3.4). Following the geometrical 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,

$$\chi_{i,j,k} = \sum_{ele} \chi_{ele},$$

(3.5)

In inviscid flows, the cut-element based interface exchange term $\chi_{ele}$ accounts for the pressure and pressure work at the element interface

$$\chi_{ele} = \begin{bmatrix} 0 \\
p_{ele}^{\Gamma} \Delta \Gamma_{ele} n^\Gamma_{ele} \\
p_{ele}^{\Gamma} \Delta \Gamma_{ele} n_2^{\Gamma_{ele}} \\
p_{ele}^{\Gamma} \Delta \Gamma_{ele} n_3^{\Gamma_{ele}} \\
p_{ele}^{\Gamma} \Delta \Gamma_{ele} (n^{\Gamma_{ele}} \cdot u^{\Gamma_{ele}}) \end{bmatrix},$$

(3.6)

where $\Delta \Gamma_{ele}$ is the element interface area, $n^{\Gamma_{ele}} = [n_1^{\Gamma_{ele}}, n_2^{\Gamma_{ele}}, n_3^{\Gamma_{ele}}]$ is the element unit normal vector obtained directly from the structural interface triangle $\Gamma_{tri}$, and $u^{\Gamma_{ele}}$ is the interface velocity evaluated at the cut-element face centroid. The element interface pressure $p_{ele}^{\Gamma}$ is obtained by solving a one-sided face-normal Riemann problem with a suitable approximate or exact Riemann solver.

3.3.3. Boundary conditions for solid walls

Uncut cells in the solid part of the computational domain in the vicinity of the interface are called ghost-cells. These cells contain ghost fluid states that allow boundary conditions at the interface to be satisfied without requiring a modification of interpolation stencils in the finite-volume reconstruction scheme. For this purpose, we apply the ghost-cell methodology as originally proposed by [60] and further extended to stationary and moving boundary cut-cell methods by [61]. Finding the ghost-cells and extending
A combined FV-FE approach for FSI using a cut-cell IBM

Figure 4. Construction of the ghost-cell extending procedure for a cut-cell \((i,j,k)\).

the fluid solution over the interface does not require the fully detailed cut-cell geometry. We therefore 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 \(x_{GP}\) that contribute to the interpolation stencil of the baseline discretization are identified, see Fig. 4. Next, for each ghost-cell the boundary intercept point \(x_{BI}\) is computed such that the line segment \(x_{GP}x_{BI}\) intersects the immersed boundary in \(x_{BI}\) normal to the interface segment. The line segment is extended into the fluid region to find the image point

\[
x_{IP} = x_{BI} + n_{avg} \cdot \Delta l,
\]

where \(\Delta l = ||x_{BI} - x_{GP}||\) 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 \(\varphi_{IP}\) at the image point \(x_{IP}\):

\[
3D: \quad \varphi(x^*, y^*, z^*) = c_1 + c_2 x^* + c_3 y^* + c_4 z^* + c_5 x^* y^* + c_6 x^* z^*
+ c_7 y^* z^* + c_8 x^* y^* z^*,
\]

\[
2D: \quad \varphi(x^*, y^*) = c_1 + c_2 x^* + c_3 y^* + c_4 x^* y^*,
\]

where \(x^* = x - x_{IP}\) is the relative distance vector and \(c = \{c_i\}\) are the unknown coefficients. As shown in Fig. 4, the four (eight in 3-D) coefficients can be determined from the variable values of the four (eight in 3-D) surrounding neighboring points,

\[
c = V^{-1} \varphi,
\]

where \(\varphi\) denotes the solution at regular fluid data points and \(V^{-1}\) the inverse Vandermonde matrix, which is calculated through LU decomposition. After solving for the
inverse interpolation matrix the value at the image point is given by
\[ \varphi_{IP} = c_1 + O(\Delta^2). \]  
(3.10)

Ghost-cell values are obtained using a linear approximation along the line \( x_{GP} = x_{BI} \) that satisfies the boundary conditions at the boundary intercept location \( x_{BI} \). For Dirichlet boundary conditions, ghost-cell data are obtained as
\[ \varphi_{GP} = 2 \cdot \varphi_{BI} - \varphi_{IP} + O(\Delta^2), \]  
(3.11)

whereas Neumann boundary conditions are imposed as
\[ \varphi_{GP} = \varphi_{IP} - 2 \cdot \Delta l \frac{n^\Gamma}{\partial s} \cdot \nabla \varphi \cdot \hat{n}_{avg} + O(\Delta^2). \]  
(3.12)

3.3.4. Treatment of small cut-cells

The time step \( \Delta t \) is adjusted dynamically according to the CFL condition based on full cells of the underlying Cartesian grid. A drawback of cut-cell methods is that the fluid volume fraction of cut-cells may become arbitrarily small and therefore can lead to numerical instability with explicit time integration schemes and poor convergence with implicit methods. A stabilization of the underlying scheme is therefore required. We here employ a so-called mixing procedure, in which a cut-cell is mixed with one [47] or multiple [48, 49] surrounding cells in a conservative way. For a detailed description of the mixing procedure used in this work, see [53].

3.4. Solid: Finite Element Method

The Finite Element Method (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 (2.5) and Neumann boundary conditions (2.12) with virtual displacements \( \delta \mathbf{d} \). Subsequently, integration by parts of the divergence theorem is applied, yielding
\[ \int_{\Omega_S} \rho_S \ddot{\mathbf{d}} \cdot \delta \mathbf{d} \, dV_0 + \int_{\Gamma_{N,i}} S : \delta \mathbf{E} \, dA_0 - \int_{\Gamma_{N,i}} \hat{b}_0 \cdot \delta \mathbf{d} \, dA_0 - \delta \mathcal{W}_S^\Gamma = 0 \]  
(3.13)

with infinitesimal volume and surface elements, \( dV_0 \) and \( dA_0 \), respectively. Herein, \( \delta \mathbf{E} \) is obtained as result of the variation of the Green-Lagrange strain (2.7), i.e.
\[ \delta \mathbf{E} = \frac{1}{2} \left( (\nabla_0 \delta \mathbf{d})^T \cdot \mathbf{F} + \mathbf{F}^T \cdot \nabla_0 \delta \mathbf{d} \right) \]  
(3.14)

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 \( \delta \mathcal{W}_S^\Gamma \).

The weak form of the structural equation (3.13) is discretized in space with the FEM. Hereby, the solid domain \( \Omega_S \) is split into \( n^e \) elements \( \Omega_S^{(e)} \) (sub-domains). For details on the FEM, the reader is referred to, e.g. [62–64]. The semi-discrete weak form of the balance of linear momentum is obtained via assembly of all elementwise contributions, leading to
\[ \mathbf{M} \ddot{\mathbf{d}} + \mathbf{f}_{S,\text{int}}(\mathbf{d}) - \mathbf{f}_{S,\text{ext}}(\mathbf{d}) - \mathbf{f}_S^\Gamma = 0, \]  
(3.15)

where we have assumed the discrete virtual displacement vector \( \delta \mathbf{d} \) to be arbitrary. The vectors \( \mathbf{d} \) and \( \ddot{\mathbf{d}} \) describe the discrete acceleration and displacement vectors, respectively, \( \mathbf{M} \) denotes the mass matrix, \( \mathbf{f}_{S,\text{int}} \) and \( \mathbf{f}_{S,\text{ext}} \) the internal and external force vectors.
The interface traction of the fluid on the structure is described by $\Gamma_S$. For time integration, the generalized trapezoidal rule (or one-step-$\theta$ scheme) is employed for the structure solver in this work. Thus, applying this scheme to the semi-discrete equation (3.15), 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 applying a Newton-Raphson method. The linearized system reads

$$K_{SS}(d^n_{i+1}) \Delta d^n_{i+1} = -r_S(d^n_{i+1})$$

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

$$d^{n+1}_{i+1} = d^n_{i+1} + \Delta d^n_{i+1}.$$  

The Newton-Raphson iteration is considered to be converged if $|r_S|_2 \leq \epsilon$ is satisfied using a sufficiently small tolerance $\epsilon$.

4. Coupling procedure

4.1. Loosely coupled partitioned FSI algorithm

The great advantage of partitioned algorithms for fluid-structure interaction problems is that they combine efficient and validated solvers for each subproblem. A well-known issue of partitioned algorithms is the added mass effect [14, 16, 18], which may lead to numerical instability and is especially prevalent in incompressible flows and for high fluid-solid density ratios $\rho_F/\rho_S > 1$. While partitioned algorithms can be made strong through the introduction of equilibrium iterations [4, 11], we here apply a loosely coupled approach since compressible flows are considered in our numerical examples and numerical instabilities were not observed.

In this paper, we use a Conventional Serial Staggered (CSS) procedure for the coupling. The CSS algorithm has been frequently used for the investigation of nonlinear transient aeroelastic problems [1, 2], large-scale FSI of shock-loaded thin-walled structures [13], and compressible flows in general [12, 65]. In Fig. 5 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 reads as follows:

1. The known structural interface displacements $d^{\Gamma;n}$ and velocities $d^{\Gamma;n}$ at time $t^n$ are used to update the cut-cells list and geometric properties on the fluid side. For this purpose, the structural interface is triangulated (see Fig. 3) and subsequently the cut-element algorithm is applied.

2. Advance the fluid in time. The evaluation of the interface exchange term (3.6) and the prescription of ghost-cell values (3.11) and (3.12) use a given structural Dirichlet boundary conditions for the interface velocities: $u^{\Gamma;n+1}_e = d^{\Gamma;n}$. An interpolation procedure is needed to transfer solid velocities to the fluid interface, see Section 4.2.1.

3. Transfer the fluid interface normal tractions $\sigma^{\Gamma;n+1}_e \cdot n^{\Gamma;n}$ 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 4.2.2.

4. Advance the structure in time while the fluid interface loads act as an additional Neumann boundary condition on the solid.
Using the structural displacement $d^{\Gamma;n}$ as a predictor for the fluid solution at time $t^{n+1}$ results in a first order accurate coupling scheme [21]. Moreover, the explicit staggering algorithm is only conditionally stable since at time level $t^{n+1}$, the continuity condition is only fulfilled for the dynamic ($\sigma^{\Gamma;n+1} \cdot n^{\Gamma;n} = P^{\Gamma;n+1} \cdot \mathbf{n}_0$) and not for the kinematic part ($u^{\Gamma;n+1} = \dot{d}^{\Gamma;n} \neq \dot{d}^{\Gamma;n+1}$), which in turn explains the violation of conservation properties of energy at the interface.

4.2. 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 as well as to a change in connectivity, which inevitably leads to a non-matching discretization of the interface ($\Gamma_F \neq \Gamma_S$). Instead of simply copying data between the two solvers, as it is common for matching grids, projections and interpolations are needed. A collection of different data mapping methods can be found in [66]. In order to enforce the kinematic and dynamic constraints on the interface as described in Section 2.3, consistent interpolation strategies are needed in both directions. Quantities such as linear momentum and angular momentum should be preserved when loads from the fluid are applied to the solid in order to obtain a conservative coupling procedure. Moreover, properties of the solid such as velocities need to be interpolated to the fluid interface in order to enforce the kinematic interface condition (2.15). A simple and efficient interpolation scheme for transferring solid quantities to the fluid interface is presented in Section 4.2.1. Our method for projecting loads from the fluid to the solid is inspired by the dual mortar method. It is briefly presented in Section 4.2.2.

4.2.1. 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 $p^{\text{ele}}_{\Gamma}$ through a Riemann solver, see (3.6). Moreover, the kinematic constraint (2.15) requires matching normal velocities at the interface. We implicitly impose this condition by setting adequate values at the ghost-cells according to (3.11), which requires the evaluation of structural velocities at each boundary intercept location $x_{\text{BI}}$. Hence, an interpolation method is needed that adequately transfers structural velocities to the fluid locations of interest. Since we may obtain several cut-elements in each fluid cell, a computationally efficient interpolation procedure is required.

Fig. 6 shows a schematic of the transfer method used in this work. On the left, a general fluid-cell cut by several structural interface triangles $\Gamma_{\text{tri}}$ is shown, resulting in a number of cut-elements $\Gamma_{\text{ele}}$. On the right, a single cut-element has been extracted together with its associated structural interface element. Nodal structural ve-
Figu're 6. Schematic of interpolation procedure used for transferring solid quantities to the fluid interface.

The velocities \( \mathbf{d}^\Gamma_i, i = 1 \ldots 3 \) are communicated to the fluid solver and serve as an input for the interpolation. Considering the location of the element centroid, the interpolation for the structural velocity reads

\[
\mathbf{u}^\circ = \frac{A_1}{A_{\text{total}}} \mathbf{d}_1^{\Gamma} + \frac{A_2}{A_{\text{total}}} \mathbf{d}_2^{\Gamma} + \frac{A_3}{A_{\text{total}}} \mathbf{d}_3^{\Gamma},
\]

(4.1)

where \( A_i, i = 1 \ldots 3 \) denote sub-areas associated to each structural vertex and \( A_{\text{total}} \) the total interface triangle area. The interpolation is linear within the triangle plane and consistent, since \( \sum_{i=1}^{3} A_i = A_{\text{total}} \).

Note that no additional search algorithm is needed to identify the associated structural interface element for a single cut-element, since this information has been already stored during the geometry computation presented earlier in Section 3.3.1. Moreover, our method does not require any projection step as it would have been the case for an averaged interface \( \Gamma_{\text{cell}} \) as obtained from a level-set based approach, making this type of data transfer robust and efficient.

4.2.2. Transfer of fluid forces to solid interface

The equilibrium of forces requires the surface tractions of fluid and solid to be equal. The coupling of non-matching interfaces between fluid and solid establishes the need for a discrete projection operator \( P_{F \rightarrow S} \) for mapping the nodal fluid loads \( \mathbf{f}_F^\Gamma \) onto the nodal structural forces \( \mathbf{f}_S^\Gamma \), leading to

\[
\mathbf{f}_S^\Gamma = P_{F \rightarrow S} \mathbf{f}_F^\Gamma.
\]

(4.2)

Due to the changing connectivity of the interface on the fluid side, it is necessary to compute \( P_{F \rightarrow S} \) in each time step. Consistent and efficient coupling of non-matching interfaces with mortar methods in the context of finite element analysis has been introduced in [67, 68] and extended to fluid-structure interaction problems and fluid flow, see [69] and [70], respectively. The mortar method demands the choice of a so-called slave and master side of the interface \( \Gamma_{\text{sl}} \) and \( \Gamma_{\text{ma}} \), respectively. Primary coupling variables, such as displacements, 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 and the solid to be the master side with respect to mortar coupling.
Due to the fact that an immersed interface method for the fluid is used in our approach and the reconstruction of the interface is necessary in each time step, we only perform the projection of tractions using the mortar method. In the following, we briefly summarize the construction of the discrete projection operator $P_{F \to S}$.

Although we are not directly interested in the virtual work term
\[ \delta W_{\text{coup}} = \int_{\Gamma} \lambda^T (\delta z^d - \delta z^m) \, d\Gamma \]
which arises in the mortar framework, this will be the starting point for our derivations of the discrete projection operator. The variation of a primary variable, that is displacements or velocities, is denoted with $\delta z$. The specific choice of $\delta z$ is not relevant as it is not included in the final transfer operator. The Lagrange multiplier
\[ \lambda = \sum_{j=1}^{n^d} \Phi_j \lambda_j \]
lives on the slave side of the interface and is interpolated using so-called dual shape functions $\Phi_j$ and the discrete nodal Lagrange multipliers $\lambda_j$. It can be identified with the unknown interface tractions. The total number of slave and master nodes is denoted with $n^d$ and $n^m$, respectively. In contrast, standard shape functions $N^d_k$ and $N^m_l$ based on Lagrange polynomials are used for the interpolation of the virtual primary variables on both sides of the interface. This leads to
\[ \delta p^d = \sum_{k=1}^{n^d} N^d_k \delta p^d_k, \]
\[ \delta p^m = \sum_{l=1}^{n^m} N^m_l \delta p^m_l, \]
where the nodal virtual primary variables are denoted with $\delta p^d_k$ and $\delta p^m_l$. The special choice of the dual shape functions comes from a so-called biorthogonality condition, viz.
\[ \int_{\Gamma} \Phi_j N^d_k \, d\Gamma = \delta_{jk} \int_{\Gamma} N^d_k \, d\Gamma, \]
where $\delta_{jk}$ is the Kronecker delta. Inserting (4.4) – (4.6) into (4.3) leads to
\[ \delta W_{\text{coup}} = \sum_{j=1}^{n^d} \sum_{k=1}^{n^d} \lambda_j^T \left( \int_{\Gamma} \Phi_j N^d_k \, d\Gamma \right) \delta p^d_k - \sum_{j=1}^{n^d} \sum_{l=1}^{n^m} \lambda_j^T \left( \int_{\Gamma} \Phi_j N^m_l \, d\Gamma \right) \delta p^m_l. \]

Therein, nodal blocks of the two mortar integral matrices commonly denoted as $D$ and $M$ can be identified. This leads to the following definitions:
\[ D[j,k] = D_{jk} I_3 = \int_{\Gamma^d} \Phi_j N^d_k \, d\Gamma I_3 = \delta_{jk} \int_{\Gamma^d} N^d_k \, d\Gamma I_3, \]
\[ M[j,l] = M_{jl} I_3 = \int_{\Gamma^m} \Phi_j N^m_l \, d\Gamma I_3 \]
with the $3 \times 3$ identity tensor $I_3$, whose size is determined by the number of variables to be coupled for each node. Herein, $D$ is a square $3n^d \times 3n^d$ matrix, which has only diagonal entries due to the biorthogonality condition of the dual shape functions, whereas the
A combined FV-FE approach for FSI using a cut-cell IBM

Figure 7. Interpolation of state variables. (a) Finite Volume Method: constant value per cell, (b) Finite Element Method: linear Lagrange polynomials, (c) Discontinuous Galerkin like interpolation.

definition of \( M \) generally yields a rectangular matrix of dimensions \( 3n^{sl} \times 3n^{ma} \). The actual numerical integration of the mortar integrals can be performed in several ways, e.g. segment-based, see [71–73], or element-based, see [74–76]. Due to its superior numerical efficiency, element-based integration is used exclusively in this work.

Finally, the nodal coupling forces can be expressed using the coupling matrices and the Lagrange multipliers as

\[
\begin{align*}
 f^T_F &= D^T \lambda, \\
 f^T_S &= -M^T \lambda,
\end{align*}
\]

which can be reordered to obtain the discrete projection operator for transferring loads from the fluid to the solid. The final transfer operation does not explicitly solve for the Lagrange multipliers, but instead reads

\[
 f^T_S = -\left( D^{-1} M \right)^T f^T_F. 
\]

Hence, the discrete projection operator in (4.2) is

\[
P_{F \rightarrow S} = -\left( D^{-1} M \right)^T. 
\]

It shall be noted that the special choice of the dual shape functions leads to a diagonal \( D \) matrix and therefore, the inversion of \( D \) is a trivial task at negligible cost. It can be concluded that the tractions can be transferred from the fluid to the solid in such a way that linear and angular momentum are conserved without the need of solving a possibly large linear system when some basic ideas and concepts of mortar methods are exploited.

Special attention has to be given to the representation of the interface on the fluid side because the underlying discretization is based on the FVM. The slave side of the coupling interface consists of all cut-elements in the fluid interface. Since a cut-element may be of polygonal shape, it is triangulated for setting up the coupling matrices. In the FVM, the values of the state variables are assumed to be constant across a single cell. This is also the case for cut-elements and their triangulated subelements, see Fig. 7(a) which depicts three cut-elements. In order to represent such a piece-wise constant field, e.g. the fluid tractions acting on the solid, with finite elements, on which the proposed mortar methods are based, it is necessary to allow for jumps between elements, which is usually not possible, see Fig. 7(b). The use of \( P_0 \) finite elements would allow for a piece-wise constant field representation but this element type is due to implementation
issues not available. Instead, inspired by Discontinuous Galerkin methods, see e.g. [77], which are known to exhibit this particular feature, nodes of the triangular elements in the fluid interface are copied in order to allow each triangle to carry its own set of nodes. This leads to nodes of neighboring elements lying at the exact same position but having different nodal values for the properties in each element, see Fig. 7(c). Hence, the number of slave nodes \( n_{sl} \) is always three times the number of triangular fluid elements in the interface.

To summarize, we enable an exact representation of the piece-wise constant traction field from the immersed interface method although using first-order finite elements by duplicating the nodes in the coupling procedure. In contrast, the master side of the mortar coupling, which contains the solid interface elements, is set up with standard finite elements which are obtained based on their trace space relationship with the underlying solid discretization.

5. Validation of the FSI algorithm

In the following we present a validation of our method for rigid solids and deformable structures. The coupling time step for all simulations is chosen to be the same for both sub-domains (\( \Omega_F, \Omega_S \)) and is based on the CFL-condition for the fluid flow.

5.1. Shock wave impact on rigid cylinder

The following test case for rigid body motion has been originally proposed by [45] and has been widely adopted in the literature, see e.g. [27, 78–81]. 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 \((x, y) = (0.15, 0.05) \text{ m} \), and subsequently driven and lifted upwards by a \( \text{Ma} = 3 \) shock wave entering the domain from the left. The pre-shock conditions \( \rho_{F,R} = 1 \text{ kg/m}^3, \rho_r = 1 \text{ Pa}, u_r = 0 \text{ m/s} \) hold for \( x \geq 0.08 \text{ m} \) while for \( x < 0.08 \text{ m} \) 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 prescribed. The fluid domain is rectangular with dimensions \( 1 \text{ m} \times 0.2 \text{ m} \) and is discretized with 1600 \times 320 cells in streamwise and wall-normal direction, respectively. This leads to a grid resolution of \( \Delta x = \Delta y = 6.25 \times 10^{-4} \text{ m} \). 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 \( r = 0.05 \text{ m} \) 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. 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 more emphasis on convergence properties of the proposed coupling algorithm.

Instantaneous pressure contours at \( t = 0.14 \text{ s} \) and \( t = 0.255 \text{ s} \) are shown in Fig. 8. With respect to the cylinder position and the resulting shock patterns our results agree well to those presented by [80] and [27]. We observe a strong vortex under the cylinder, which persists over the complete cylinder trajectory, see Fig. 8, supporting the results of [27, 78]. By further increasing the mesh resolution up to \( \Delta x = \Delta y = 1.5625 \times 10^{-4} \text{ m} \) the vortex is still apparent, excluding numerical dissipation being responsible for the formation of the vortex, which is proposed by [80]. As stated by [27], a Kelvin-Helmholtz instability of the contact discontinuity present under the cylinder may cause this vortex.
A combined FV-FE approach for FSI using a cut-cell IBM

Figure 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

Figure 9. Convergence study of (a) horizontal and (b) vertical cylinder center position for different fluid mesh resolutions. (– – – –) present results, (– – – – – –) [80], (– – – – – –) [27]. x-axis is given in logarithmic scale.

Fig. 9(a)/(b) shows convergence results on the final horizontal and vertical position of the center of mass of the cylinder together with results from literature [27, 80]. The final position is in the same range as in [27, 80]. Our results show a fast convergence rate, similar to the results obtained by [27].

5.2. Shock wave impact on deforming panel
The behavior of a cantilever panel subjected to a shock tube flow is analyzed in the following. This test case has been investigated both experimentally and numerically by the Institut Universitaire des Systèmes Thermiques Industriels (IUSTI) laboratory [12]. The experimental setup, as shown in Fig. 10, consists of a deformable panel of length $l = [40, 50]$ mm and width $b = 1$ mm placed within a shock tube. The panel is hit by
a Ma = 1.21 shock wave, which enters the domain from left. The panel is made of steel ($E_S = 220$ GPa, $\rho_{S,0} = 7600$ kg/m$^3$, $\nu_S = 0.33$) and is clamped to a non-deformable forward-facing step at its lower end. The pre-shock conditions resemble air at rest and are set to $\rho_{F,R} = 1.189$ kg/m$^3$, $p_R = 100$ kPa, $u_R = 0$ m/s, while the post-shock values are $\rho_{F,L} = 1.616$ kg/m$^3$, $p_L = 154$ kPa, $u_L = 109.68$ m/s. The fluid domain is rectangular with dimensions 0.295 m $\times$ 0.08 m in width and height. Since the problem is considered as being fully two-dimensional, we adopt a constant thickness of 0.001 m in spanwise direction. Slip wall boundary conditions are employed for all boundary patches except for the inflow patch, where we prescribe non-reflective inflow boundary conditions based on Riemann invariants [82]. 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. 11. The panel is discretized using $65 \times 2$ ($l = 50$ mm) or $55 \times 2$ ($l = 40$ mm) 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. The method of enhanced assumed strains (EAS), as introduced in [83], is used in order to avoid shear locking phenomena, which may affect
A combined FV-FE approach for FSI using a cut-cell IBM

Fig. 12. Time evolution of panel tip displacement for 50 mm panel length using different fluid meshes. (—) Mesh A, (---) Mesh B

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. 12 shows the time evolution of the horizontal displacement at the panel tip for the 50 mm panel length case on Mesh A and Mesh B. One can observe that the panel motion is almost identical for both fluid meshes throughout the whole simulation time. Results that will be presented below are obtained on fluid Mesh B, although such a high resolution is not needed to capture the dynamics of the system.

We start with a qualitative analysis of the flow field for the 50 mm panel. Fig. 13 shows numerical schlieren (left) and experimental shadowgraph visualizations (right) extracted from [12] 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 initial normal shock undergoes transition to a cylindrical shock front due to the 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 ($t = 280 \mu s$), but are not resolved in our results due to the 2-D setup. Fig. 14 shows a numerical schlieren image at $t = 4.17 ms$, illustrating the maximum panel deflection together with the interaction of the main vortex and the upstream moving shock wave.

A more quantitative analysis is presented in Fig. 15, where the time evolution of the horizontal panel tip displacement is plotted. Fig. 15(a) refers to the 50 mm panel length case and Fig. 15(b) to the 40 mm case, respectively. In addition to experimental values [12] represented through error-bars, we include recent inviscid numerical results of [84], who employed a Finite-Element based partitioned fluid-structure interaction approach utilizing the Arbitrary Lagrangian-Eulerian (ALE) description to account for moving boundaries and coupling with Lagrangian shell elements. Moreover, numerical results by [12] are added, who assumed a two-dimensional but viscous flow in the lami-
For the 50 mm panel case, see Fig. 15(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 [12] 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 is connected 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
A combined FV-FE approach for FSI using a cut-cell IBM

![Figure 14. Contour of density gradient magnitude at \( t = 4.17 \) ms.](image)

![Figure 15. Time evolution of panel tip displacement for (a) 50 mm and (b) 40 mm panel length. (——) present results, (—−−) [12], (⋯⋯) [84]. Error bars denote experimental data [12].](image)

is found very close to the analytical period of 2.87 ms when considering the first eigenmode of a clamped plate submitted to an impulse load [12]. The experimental period is given as 3.8 ms.

Based on these uncertainties, a second case with 40 mm panel length has been studied. With the shorter panel, the stresses on the base part are reduced, which also reduces the influence of the base on the panel motion. We observe excellent agreement with experimental data and numerical references, see Fig. 15(b), which proves the assumption of a locally non-rigid base plate.

Finally, the pressure signal recorded at \( (x, y) = (0.035, 0.08) \) m for both panel lengths is compared to the same numerical and experimental database in Fig. 16. 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 findings for the 50 mm panel case, almost identical time evolution up to \( t = 2 \) ms is observed for the 40 mm panel case. After that time, the experimental pressure continuously drops due to the arrival of reflected expansion waves inside the shock tube, which are not taken into account in the numerical simulations.
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 [85]. For the setup considered here, see Fig. 17, linear instability theory predicts a critical Mach number of \( \text{Ma}_{\text{crit}} = 2.0 \) above which continuous growth of oscillations amplitudes can be observed [86]. To trigger the instability, the pressure acting on the bottom of the panel is decreased by 0.1% initially and 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 \( \text{Ma}_{\text{crit}} = 2.0 \) describes a perfect oscillation without damping or amplification [1], this test case will directly highlight the amount of numerical damping present in our algorithms. Note that in contrast to [1, 84, 87] no subcycling in the fluid is used for our simulations, removing any uncertainties originating from the adopted subcycling scheme.

We consider a supersonic inviscid flow over a flat plate that is clamped at both ends, see Fig. 17. The plate has a length of \( l = 0.5 \text{ m} \), a thickness of \( t = 0.00135 \text{ m} \), a Young’s modulus of \( E_S = 77.28 \text{ GPa} \), a Poisson’s ratio of \( \nu_S = 0.33 \), and a density of \( \rho_{S,0} = 2710 \text{ kg/m}^3 \). The structure is discretized using \( 100 \times 2 \) tri-linearly-interpolated hexahedral elements in streamwise and wall-normal direction, respectively. To avoid shear locking phenomena, the EAS method is used. If not stated otherwise, a geometric linear analysis of the structure is performed for comparison with references in the literature. The time integration factor \( \theta = 0.5 \) is chosen in order to reduce numerical damping. The fluid free-stream properties are: \( \rho_{F,\infty} = 0.339 \text{ kg/m}^3 \), \( p_{\infty} = 28 \text{ kPa} \) and \( \text{Ma} = [1.9 \ldots 2.3] \). The computational domain together with a general view of the fluid mesh resolution is shown.
A combined FV-FE approach for FSI using a cut-cell IBM

FIGURE 18. General view of the computational domain and mesh resolution. Every 5th and 20th cell is shown in the $x$- and $y$-direction, respectively.

FIGURE 19. (a) Vertical deflection of the plate at $x = 0.6\,m$ for $Ma \in [1.9, 2.0, 2.05, 2.3]$. (—) $Ma = 1.9$, (⋯⋯) $Ma = 2.0$, (⋯⋯) $Ma = 2.05$, (⋯⋯) $Ma = 2.3$. (b) Geometrically linear and non-linear plate deflections at $x = 0.6\,m$ for $Ma = 2.3$. (—) linear, (⋯⋯) non-linear. The gray shaded area indicates the initial perturbation time.

in Fig. 18, highlighting every 5th and 20th fluid cell in $x$- and $y$-direction, respectively. A total number of 204,000 cells is used for the fluid domain. The grid is uniform in the region around the panel ($0.25\,m \leq x \leq 0.75\,m$) with a cell size of $\Delta x = 2.08 \times 10^{-3}\,m$ and $\Delta y = 7.33 \times 10^{-5}\,m$. A cavity of height $2.2 \times 10^{-2}\,m$ is added below the panel ($y \leq 0\,m$) to account for the panel motion in this region. Since the problem is two-dimensional, we adopted a constant thickness of $\Delta z = 0.005\,m$ 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 leading 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 $Ma = [1.9, 2.0, 2.05, 2.3]$ is shown in Fig. 19(a). The area shaded gray marks the time period during which the pressure acting on the bottom of the panel is reduced. Exponential growth of the initial disturbance is observed for $Ma = 2.3$, which confirms analytical and numerical results [1,84,86,87]. Considering the remaining results for Mach numbers around the critical value $Ma_{\text{crit}} = 2.0$, however, we observe that all panel oscillations are damped. Further simulations are needed to precisely predict the onset of flutter with our method since the interval $Ma = [2.05, 2.3]$ is only a rough
Moreover, the reason for the discrepancy concerning the correct prediction of flutter onset between our results and analytical theory is still an open question and currently under investigation. Fig. 19(b) shows a comparison between geometrically linear and non-linear panel solutions for a Mach number of \( \text{Ma} = 2.3 \). While the linear solution predicts exponential growth of the oscillations, the geometrically non-linear case exhibits limited displacement amplitudes. According to [85] the behavior of panels 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 within the fluid domain is shown in Fig. 20(a) together with its associated flutter mode.
Figure 22. General view of the computational domain and mesh resolution. The triangulated solid interface is illustrated additionally.

with the associated flutter mode in Fig. 20(b) at the time instant \( t = 0.06746 \) s. The deflection of the panel leads to the formation of compression and expansion waves within the fluid domain. 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 peak displacement for the flutter mode is found at 70% of the panel length, confirming other numerical findings [1, 84, 87].

5.4. Buckling of a three-dimensional inflated membrane

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 membrane and a \( \text{Ma} = 1.21 \) shock wave. The pre- and post-shock fluid state are equal to the conditions introduced in Section 5.2, whereas the initial shock position is located at \( x = -0.05 \) m. Geometric details are shown in Fig. 21. The spherical membrane has a thickness of \( t = 0.001 \) m and an inner radius \( r_i = 0.029 \) m with its center \( M \) located at \((x, y, z) = (0, -0.005, 0) \) m. Its material properties are \( E_S = 0.07 \) GPa, \( \rho_{S,0} = 1000 \) kg/m\(^3\) and \( \nu_S = 0.35 \) for the Young's modulus, the density and the Poisson ratio, respectively. The membrane 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 \( p_R \). Zero displacements in all three directions
Figure 23. Pressure signal recorded at sensor position \((x, y, z) = (0, 0.04, 0)\) m.

Figure 24. Pressure distribution together with velocity vectors at different time instances. Color scale ranges from white to black using 20 equally spaced contour levels for \(p \in [220 - 260] \text{kPa}\).
A combined FV-FE approach for FSI using a cut-cell IBM

![Images of spherical structures with labeled points A and B at different time steps: (a) t = 0 ms, (b) t = 0.285 ms, (c) t = 0.7704 ms, (d) t = 1.22 ms, (e) t = 1.4 ms, (f) t = 1.5 ms.]

**Figure 25.** Development of the magnitude 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 $|e_2| \in [0 - 0.13]$. Monitoring points A and B are marked with squares.

Monitoring points A and B are marked with squares.

are prescribed for structural nodes located at the bottom of the membrane at $y = 0$ m. The time integration factor $\theta = 0.5$ is chosen.

Fig. 22 depicts the computational domain in conjunction with 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.
at all boundaries except for the inflow, where all flow quantities are prescribed leading to a fully reflective boundary condition. In the region around the membrane, a uniform grid is used with cell sizes $\Delta x = \Delta y = \Delta z = 0.001 \text{ m}$. 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. 23, the pressure signal recorded at the sensor position $P$ with $(x, y, z) = (0, 0.04, 0) \text{ m}$ is depicted. The jumps at approximately $t = 0.1167 \text{ ms}$ and at $t = 1.2196 \text{ ms}$ mark the times when the shock wave passes the sensor. Pressure distributions and velocity vectors at the $xy$-midplane are shown in Fig. 24 for different time instances. The corresponding strain distributions in the thin-walled membrane are presented in Fig. 25. The magnitude of the Euler-Almansi strain tensor $|\varepsilon|_{2} = \sqrt{\varepsilon : \varepsilon}$ evaluated at each element center 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. 24(a) and Fig. 25(a). Due to the overpressure induced by the shock, Fig. 24(b), the windward side of the membrane is compressed, see Fig. 25(b), and is subsequently bouncing back due to its elastic behavior. At time $t = 0.7704 \text{ ms}$, buckling of the thin-walled membrane occurs at its tip, deflecting the flow as depicted in Fig. 24(c). The displacement of the tip node at initial position $(x, y, z) = (0, 0.025, 0) \text{ m}$ (monitoring point A) is given in Fig. 26(a): the $y$-deflection is approximately 1.5 mm during this first shock induced dimpling process. As the shock hits the membrane after reflection at the end wall, see Fig. 24(d)-(f), the pressure increases 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. 24(d)-(f) and Fig. 25(d)-(f). At $t = 1.5 \text{ ms}$, the magnitude of the Euler-Almansi strain in the most distorted regions rises up to 0.127, see Fig. 25(f). Considering the monitoring point B, which is initially located at $(x, y, z) = (0.011912, 0.020912, 0.009308) \text{ m}$ in one of the dimples, a total deflection of 5.23 mm is found, see Fig. 26(b).

6. Summary and conclusions

We presented a mixed Finite Volume – Finite Element coupling approach for the interaction between a compressible fluid and a deformable structure. The approach 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
A combined FV-FE approach for FSI using a cut-cell IBM

procedure for coupling of the two domains. This loose coupling method did not show any numerical instabilities during the investigation of the presented test cases, which confirms the suitability of the approach for compressible flow environments.

An exact representation of the interface within the fluid domain is achieved by means of a cut-element based Immersed Boundary method. This method strictly enforces geometric conservation and prevents the formation of numerical artifacts, such as spurious pressure oscillations that were observed with the former level-set based geometry reconstructions near the conjoined interface. An appropriate approximation of a bilinear structural surface element, which emerges from the tri-linearly interpolated geometry elements of the structural problem, is obtained via division into four triangles using the vertices and the geometric center of the element.

The presented framework inevitably leads to a non-matching discretization of the interface between both sub-domains, hence demanding for an efficient and conservative data transfer. We employ a simple and efficient interpolation procedure for each cut-element for transferring velocities from the solid to the fluid interface based on a linear interpolation within the hosting structural interface triangle. No additional search algorithm is needed, since each cut-element stores its associated structural element during the geometry computation. Moreover, no projection step is needed since we resort to the exact representation of the triangulated interface. Pressure forces are transferred to the solid interface in a conservative way with a discrete projection operator, which is based on concepts used in mortar methods. We do not solve for the Lagrange multipliers because they cancel out during derivation of the projection operator. Nevertheless, the use of dual shape functions for the Lagrange multipliers makes this method very efficient. Since each cut-element represents a piece-wise constant fluid load, it is necessary for an appropriate representation within the mortar framework to allow for jumps between the finite elements. For this purpose we borrow ideas from Discontinuous Galerkin methods where each element carries its own, and independent, set of node values.

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 shock-loaded rigid and deformable structures. Some discrepancies were observed with respect to the correct prediction of flutter onset, which is subject of ongoing investigations. The ability of our method to handle three-dimensional FSI problems involving large and complex structural deformations has been demonstrated through a numerical example consisting of a flexible inflated membrane interacting with a shock wave.

Future work will focus on large-eddy simulations of reference configuration D.A2 provided by TP D6. The experimental setup consists of a pitching shock generator, whose incident shock wave impinges on a flexible panel and interacts with a fully turbulent boundary layer.

Acknowledgments

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).
References


[48] Pember, R. B., Bell, J. B., Colella, P., CurTchfield, W. Y. AND WELCOME,
A combined FV-FE approach for FSI using a cut-cell IBM


