Transient fluid–structure interaction with non-matching spatial and temporal discretizations

R. Jaiman a,⁎, P. Geubelle b, E. Loth b, X. Jiao c

a Altair Engineering, Inc., Mountain View, CA 94043, USA
b Department of Aerospace Engineering, University of Illinois, Urbana, IL 61801, USA
c Department of Applied Mathematics and Statistics, Stony Brook University, NY 11794, USA

ABSTRACT

This paper presents a review of spatial and temporal discretization schemes for unsteady flow interacting with structure. Two types of spatial coupling schemes are analyzed: (i) point-to-element projection and (ii) common-refinement based projection. It is shown that the point-to-element projection schemes may yield inaccurate load transfer from the source fluid mesh to the target solid mesh, leading to a weak instability in the form of spurious oscillations and overshoots in the interface solution. The common-refinement scheme resolves this problem by providing an accurate transfer of discrete interface conditions across non-matching meshes. With respect to the temporal discretization, three coupling techniques are assessed: (i) conventional sequential staggered (CSS); (ii) generalized serial staggered (GSS) and (iii) combined interface boundary condition (CIBC). Traditionally, continuity of velocity and traction along interfaces are satisfied through algebraic interface conditions applied in a sequential fashion, which is often referred to as staggered computation. In existing partitioned staggered procedures, the interface conditions may undermine stability and accuracy of coupled fluid–structure simulations. By utilizing the CIBC technique on the velocity and traction boundary conditions, a staggered coupling procedure can be constructed with similar order of accuracy and stability as standalone computations. The effectiveness of spatial and temporal coupling schemes is investigated with the aid of simple 1D examples and new 2D subsonic flow-shell aeroelastic simulations.

© 2011 Elsevier Ltd. All rights reserved.

1. Introduction

This work is motivated by the need to couple unsteady fluid solvers to dynamic structural models for efficient and accurate predictions of aeroelastic instabilities [1,2]. Although substantial progress has been made in the development of aeroelastic solvers, further improvements in efficiency and accuracy must be pursued to allow time–accurate nonlinear aeroelastic simulations to truly influence the design cycle. An essential feature of such a simulation is the accurate description of the deformable fluid–structure interface and thus the characterization of the fluid flow in a domain with moving boundaries. The simulation of this class of problems is generally accomplished by partitioned staggered schemes [3–6] using a Lagrangian finite element formulation for the structural domain and an Arbitrary Lagrangian Eulerian (ALE) finite volume formulation [7] for the fluid domain. Partitioning refers to the separate computational treatment of each subdomain at a given time level. Staggering refers to sequential interface-driven time integration of the fluid and structure equations. Staggering is generally achieved with the aid of velocity continuity (Dirichlet) and traction (Neumann) interface conditions.

The partitioned staggered approach presents two key challenges. The first one deals with momentum and motion transfer between differing mesh representations of the fluid and the solid subdomains. Such meshes are in general nonmatching (i.e., have disparate nodal connectivities and positions along the interface) because of differing mesh resolution requirements or discretization schemes. For example, the fluid system usually requires a finer mesh to capture nonlinear features such as shock waves and boundary layers. In addition, different types of elements used by the solvers and discretization errors in the respective subdomain geometries can also cause mesh mismatch on the interface. Three issues are with the spatial coupling methods across nonmatching meshes [8,9]: (a) How to couple two different fluid and structural solvers in space? (b) How to satisfy interface continuity conditions in an accurate and energy conservative way across flat and curved interfaces? (c) What are the effects of point-to-element and element-to-element mapping in transferring momentum and energy across nonmatching interface meshes? In [8,9], closed-form error indicators were proposed to assess and compare the accuracy and
conservation of various spatial coupling methods. These load transfer schemes were then applied to the simulation of transient FSI problems of increasing complexity, allowing for a direct comparison on the effect of the coupling schemes on the local and global features on the numerical solution. These various load transfer schemes are summarized in the present paper, which also provides additional test problems.

The second challenge is concerned with the stability, accuracy and convergence of staggered time-integration schemes used for the partitioned approach [10,11] to couple the separate time integrators of fluid and structure. With regard to the temporal coupling, a number of partitioned staggered methods have been proposed to solve FSI problems and can be categorized as either strongly-coupled [4,11–13] or loosely-coupled [2,3]. Strongly-coupled methods typically involve predictor–corrector sub-iterations to ensure the convergence of the interface properties at each coupling time step. However, the associated iterations increase the complexity of the implementation of FSI analyzes as well as the computational cost at each time step. Explicit loosely-coupled staggered methods integrate the fluid and the structure equations once and independently at every time step. These are often the methods of choice due to their relative simplicity and low computational cost. However, these procedures often suffer from numerical instability and temporal inaccuracy caused by spurious energy production along the interface due to the time lag [10,11], and special treatments are generally required to address these issues. In this paper, new forms of the interface conditions are constructed for the consistency of residual interface equations. These forms are used herein to establish a new coupling method, referred to as the Combined Interface Boundary Condition (CIBC) scheme. This scheme imposes higher-order interface corrections based on the coupled differential equations so as to counteract the artificial energy production caused by the staggering process. This is in contrast to the algebraic velocity and traction continuity conditions enforced sequentially along the interface in conventional coupling schemes such as the CSS scheme.

The main contribution of this work is to design a family of consistent, stable and accurate fluid–structure methods. After briefly summarizing loosely-coupled partitioned approach in Section 2, the spatial discretization is provided in Section 3. The temporal coupling methods are presented in Section 4. Finally, we demonstrate and assess in Sections 5 and 6 the effectiveness of the coupling methods in FSI simulations of increasing complexity that include a new 2D subsonic flow-shell problem.

2. Loosely-coupled partitioned method

The coupled FSI equations comprise the initial-boundary value problems of the fluid and the structure, complemented by the traction (dynamic) and displacement (kinematic) boundary conditions at the fluid–structure interface. In the partitioned approach, the decomposed fluid and solid domains share a common interface boundary. The structural equations are conventionally formulated in Lagrangian coordinates on a mesh that moves along with the material, while the fluid equations are formulated in Eulerian coordinates, where the mesh serves as a fixed reference for the fluid motion. For the coupling of these media, the fluid solutions are accommodated with the structural motion by an arbitrary-Lagrangian-Eulerian formulation [7]. Particular care must be taken to properly couple the fluid and the structure along the interface between the media. The coupling of the fluid and structural response can be achieved numerically in different ways, but the interface conditions of displacement continuity (motion transfer) and traction equilibrium (momentum transfer) along the fluid–structure surfaces must be satisfied.

At each FSI cycle, two interface boundary conditions corresponding to the continuity of traction and velocities must be satisfied along a common interface boundary \( \Gamma = \partial \Omega_f \cap \partial \Omega_s \). Let \( P_f \) and \( u_f \) denote the fluid traction and displacement fields along the fluid interface \( \Gamma_f \), while \( P_s \) and \( u_s \) denote the structural side traction and displacement fields along the interface \( \Gamma_s \), respectively. The equilibrium of momentum and compatibility of velocity field can be expressed as

\[
P_f = P_s, \quad \frac{\partial u_f}{\partial t} = \frac{\partial u_s}{\partial t} \quad \text{on } \Gamma,
\]

where \( P_f = \rho \nabla u_f - \sigma_n u_f \) and \( P_s = \sigma_s \nabla u_s \). Here, \( \rho \) is the fluid pressure along the interface; \( \sigma_s \) and \( \sigma_n \) are the fluid viscous tensor and structure stress tensor, respectively; \( \nabla \) and \( \nabla_{sd} \) are the unit outward local normals along the fluid interface \( \Gamma_f \) and structure interface \( \Gamma_s \), respectively, with \( \nabla = -\nabla_{sd} \). For inviscid flows, the velocity continuity is replaced by the slip-wall boundary condition

\[
\frac{\partial u_f}{\partial t} \nabla_{sd} = - \frac{\partial u_s}{\partial t} \nabla_{sd} \nabla_{sd} u_s,
\]

and the fluid traction vector only includes the effect of the pressure field, i.e., \( P_f = \rho \nabla u_f \).

3. Spatial discretization: load transfer schemes

Although nonconservative interpolation-based schemes have been proposed for FSI applications [14], conservative transfer schemes are considered as conservation has been found to be crucial for the repeated data transfer of large spatial gradients (e.g., shocks) [3] and for the stability of partitioned methods [10].

In general, conservative load transfer schemes used for generic non-matching interfaces can be categorized in three distinct classes: point-to-point, point-to-element and common-refinement schemes. In the class of point-to-point schemes, there have been recent developments based upon meshless (kernel-based) methods, which perform interpolation using positive definite and radial basis functions [6]. These meshless methods allow to couple structural and fluid domains by reducing them to pure point information. The schemes based on point-to-point interpolations are not evaluated in this paper, which focuses on the conservative point-to-element and common refinement schemes described in the next section.

3.1. Point-to-element projection schemes

Two point-to-element projection schemes are commonly used in FSI, the node projection (N-P) and the quadrature projection (Q-P), as discussed in [8]. First proposed by Farhat et al. [15], the N-P scheme projects the fluid nodes onto the solid surface element to extract the load vector on the solid interface nodes. Let \( N_f \) and \( N_s \) denote the standard finite element shape functions associated with node \( i \) of the fluid and node \( j \) of the solid interface mesh, respectively, and \( \mathbf{t}_l \) and \( \mathbf{t}_l^s \) the approximate tractions at the corresponding nodes. Let \( \mathbf{R}_l = \int_{\Gamma_l} t d\Gamma \) denote the fluid load vector and \( \mathbf{R}_l^s = \int_{\Gamma_l} t d\Gamma^s \) the solid load vector. In particular, the extracted load vector on the solid surface node is then

\[
\mathbf{R}_l^s = \sum_{i=1}^{m} N_f^i(x') \mathbf{R}_l^i,
\]

where \( x' \) denotes the location of node \( i \) of the fluid interface mesh.

Similar to the N-P approach, the Q-P scheme proposed by Cebral and Löhner [3] projects the fluid quadrature points onto the solid surface element to extract \( \mathbf{t}_l \) and \( \mathbf{R}_l \) on the solid boundary. Specifically, \( \mathbf{R}_l \) is evaluated by

\[
\mathbf{R}_l = \sum_{j=1}^{c} N_s^j \mathbf{t}_l^s d\Gamma^s,
\]

where \( c \) denotes the number of quadrature points.
where \( e_i \) denotes the number of elements on the fluid interface mesh and \( \sigma_i \) its \( i \)th element. It can be considered as a rough discretization of the Galerkin weighted residual method, where the integration is performed piecewise over the fluid interface elements with numerical quadrature. To avoid the situation in which a solid target element would receive no load, this scheme may rely on an adaptive quadrature rule by adaptively increasing the number of quadrature points. Further details about this load transfer scheme can be found in [3].

Both point-to-element transfer schemes are conservative but do not pass the traction continuity test due to the inaccurate transfer of loads or tractions from the source fluid mesh to the target solid mesh, as illustrated by Jaiman et al. [8] by a constant loading on one mesh and transferring it to another mesh. Hence, such schemes generally lead to local errors for non-matching meshes while satisfying the conservation of energy across the distinct interface meshes.

3.2. Common-refinement projection scheme

This scheme defines a common refinement on a reference surface \( \Gamma_r \) of the fluid–solid interface. In the finite element form, the spatial configuration of the fluid and solid interface meshes can be parameterized as

\[
\mathbf{x}_i \approx \sum_{i=1}^{m} N_i^f(x) \mathbf{x}^f_i \quad \text{on} \quad \Gamma^f, \quad \mathbf{x}_i \approx \sum_{j=1}^{m} N_j^s(x) \mathbf{x}^s_j \quad \text{on} \quad \Gamma^s. \tag{5}
\]

In the 2D common-refinement transfer formulations [8], the reference surface \( \Gamma_r \), can be computed by averaging the input meshes on \( \Gamma^f \) and \( \Gamma^s \), where the superscript \( ^b \) denotes the approximation using finite elements.

In general, the topology of the common refinement is defined by the intersection of the elements of input meshes, or the “subelements.” There can be various situations of intersecting of elements \( \sigma_j \) and \( \sigma_i \) over \( \Gamma^f \) and \( \Gamma^s \), respectively. For example, in order to determine whether the two elements \( \sigma_j \) and \( \sigma_i \) intersect, we loop over all the nodes of \( \sigma_j \). The two elements intersect if we can find at least one node of \( \sigma_j \) that lies inside \( \sigma_i \). This can be easily achieved with the aid of the mapping of global to local coordinates available in any finite element code. These subelements are illustrated in Fig. 1 for the 2D case. The implementation of the common refinement scheme to more complex 3D cases is described in [16].

In the common-refinement scheme, the load vector \( \mathcal{R}_i \) over the common refinement nodes is computed as

\[
\mathcal{R}_i = \sum_{i=1}^{e} \sigma_i \mathbf{t}_i \mathbf{d} \Gamma, \tag{6}
\]

which can be derived from a least-squares minimization [9], where \( e_i \) denotes the number of subelements on the common refinement, and \( \sigma_i \) denotes its \( i \)th subelement. The integration point locations and their weight functions can be determined based on the Gaussian quadrature integration. As shown in Section 5, the choice of the load transfer scheme can substantially affect the precision of the FSI simulation results, with the common refinement scheme offering substantial improvement in accuracy compared to the N-P and Q-P schemes.

4. Temporal discretization: partitioned staggered schemes

4.1. Conventional sequential staggered method

A simple and popular partitioned procedure for solving FSI problems is the conventional sequential staggered (CSS) scheme [5] whose generic cycle is described in Fig. 2a. The basic steps of the CSS cycle is presented in Algorithm 1 in the semi-discrete form between the time interval \([t^n, t^{n+1}]\) with time step size \( \Delta t = t^{n+1} - t^n \), where \( n \) corresponds to the staggered coupling time level.

In Algorithm 1, \( \alpha \) and \( \beta \) are weighting parameters to interpolate the structural velocity and fluid traction solutions between the time levels \( n \) and \( n + 1 \). In the case of \( \alpha = 1 \), the solutions at time level \( n + 1 \) are ignored and the previous solution at time step \( n \) is transferred to the respective subdomains. This method is often referred to as parallel conventional staggered method [5], since the fluid and structural subdomains can start their computations at the same time level and perform their inner subdomain integrations in a parallel way. When \( \alpha = 0 \) and \( \beta = 0 \), the CSS method as shown in Fig. 2a is obtained.

Although the CSS scheme appears attractive from a computational viewpoint, it may suffer significantly from destabilizing effects introduced through the interface discretization [11,10]. In the CSS scheme, the one-sided approximation to the traction and velocity at the interface can be inaccurate in highly nonlinear transient simulations. More precisely, the numerical treatment of boundary conditions create inconsistencies in omitting and retaining some terms on the neighboring discrete space and time slabs along the interface. Depending on the direction of the interface acceleration, high under- or over-prediction of pressure may occur.
due to the lack of energy equilibrium across the fluid–structure interface.

In other words, any small error in the interface displacements imposed onto the fluid by the structure may result in large errors in fluid pressure. In general, due to a time lag between the structural and the fluid computations, the sequential staggered solution techniques are at most first-order energy accurate due to $O(\Delta t)$ in time on $\Gamma$ [17].

Algorithm 1. Conventional sequential staggered (CSS) scheme.

1. Start from known solutions in both subdomains
2. Each FSI cycle between $t \in [t^n, t^{n+1}]$
   (a) Apply Dirichlet velocity continuity condition by imposing structural motion
      \[ \mathbf{v}_s^{n+1} = (1 - \varepsilon) \mathbf{v}_s^{n+1} + \varepsilon \mathbf{v}_f^{n+1} \] on $\Gamma$, where $\varepsilon \in [0, 1]$
   (b) Set $\mathbf{x}_f^{n+1} = \mathbf{u}_f^{n+1}$ and update fluid dynamic mesh
      \[ \nabla \cdot [(1 + \tau_m) \nabla \mathbf{x}_f^{n+1}] = 0 \] in $\Omega_f$
      $\mathbf{x}_f^{n+1}(t)$ represents the time-varying velocities
      $\tau_m$ is a mesh stiffness variable

c. Solve ALE fluid equations for new conservative variables $\mathbf{w}_f$
   \[ \frac{d}{dt} (\mathbf{Q}_f^{n+1} \mathbf{w}_f^{n+1}) = -\mathbf{F}(\mathbf{w}_f^{n+1}, \mathbf{x}_f^{n+1}, \mathbf{n}_f^{n+1}) \] in $\Omega_f$
   where $\mathbf{w}_f$ represents the vector of conserved mass, momentum and energy quantities
   $\mathbf{F}(\mathbf{w}_f, \mathbf{x}(t) \mathbf{x}_f(t), \mathbf{n}_f(t))$ denotes the discrete fluxes in the ALE form
   $\mathbf{n}_f(t)$ surface normals to the control volume boundary faces
(d) Extract new interface traction $\mathbf{P}_s^{n+1}$ and apply Neumann condition
   \[ \mathbf{P}_s^{n+1} = (1 - \beta) \mathbf{P}_s^{n+1} + \beta \mathbf{P}_s^n \] on $\Gamma$, where $\beta \in [0, 1]$
(e) Solve structure equation using known interface traction $\mathbf{P}_s^n$
   \[ M_s \frac{d^2 \mathbf{u}_s^{n+1}}{dt^2} = -\mathcal{R}_s^{eff}(\mathbf{P}_s^n, \mathbf{u}_s^{n+1}) \] in $\Omega_s$
   where $M_s$, $\mathbf{u}_s$, $\mathcal{R}_s^{eff}$ are the global mass matrix, displacement vector, effective load vector

4.2. Generalized serial staggered method

As proposed in [18,19], the temporal accuracy of the staggered partitioned scheme can be greatly improved by applying a structural prediction step based on higher-order interface velocity extrapolation and momentum averaging. There are several variants for the selection of the structural extrapolation and momentum averaging relations on the basis of discrete energy arguments [20,21].

Algorithm 2. Generalized serial staggered (GSS) method.

1. Start from known solutions in both subdomains
2. Each FSI cycle between $t \in [t^n, t^{n+1}]$
   (a) Predict interface velocity using (7) for time station $t^{n+1}$
      \[ \mathbf{v}_s^{n+1} = \mathbf{v}_s^n + \Delta t \mathbf{v}_s^{n+1} - \alpha \mathbf{v}_s^n \] on $\Gamma$
   (b) Set $\mathbf{x}_f^{n+1} = \mathbf{u}_f^{n+1}$ and update fluid dynamic mesh
      \[ \nabla \cdot [(1 + \tau_m) \nabla \mathbf{x}_f^{n+1}] = 0 \] in $\Omega_f$
   (c) Advance ALE fluid domain for new conservative variables $\mathbf{w}_f$ and compute a new traction vector
      \[ \frac{d}{dt} (\mathbf{Q}_f^{n+1} \mathbf{w}_f^{n+1}) = \mathbf{Q}_f(\mathbf{w}_f^{n+1}, \mathbf{x}_f^{n+1}, \mathbf{x}_f^{n+1}) \] in $\Omega_f$
   (d) Compute interface traction using the linear combination formula (8)
      \[ \mathbf{P}_s^{n+1} = (1 - \beta) \mathbf{P}_s^{n+1} + \beta \mathbf{P}_s^n \] on $\Gamma$
   (e) Solve structure equation using known corrected traction $\mathbf{P}_s^{n+1}$
      \[ M_s \frac{d^2 \mathbf{u}_s^{n+1}}{dt^2} = \mathcal{R}_s^{eff}(\mathbf{P}_s^{n+1}, \mathbf{u}_s^{n+1}) \] in $\Omega_s$

One representative variant is the generalized serial staggered (GSS) method given by Algorithm 2 [21]. The interface velocity $\mathbf{v}_s^{n+1}$ can be predicted according to the structural surface
extrapolation

\[ \mathbf{v}^{n+1}_p = \mathbf{v}^n + \Delta t (x_0 \mathbf{v}^n - \mathbf{v}^{n+1}_p), \]

(7)

where the real constants \( x_0 = 3/2 \) and \( x_1 = 1/2 \) are needed to yield second-order accuracy. The structural prediction is trivial if \( x_0 = x_1 = 0 \).

The improved momentum or load averaging formula is given by

\[ \mathbf{P}^{n+1}_k = (1 - \beta) \mathbf{P}^{n+1}_s + \beta \mathbf{P}^{n}_s, \]

(8)

with the constant \( \beta = 1/2 \), where the subscript P and C emphasize the prediction and correction, respectively.

These interface formulations are shown to preserve second-order accuracy of the coupled simulations using Taylor series truncation arguments [21]. This structural prediction-based staggered algorithm [20], named the Improved Serial Staggered (ISS) procedure, can also be constructed in a leap-frog manner where the fluid computation is performed at a non-collocated time level \( t^{n+1/2} \) and structural solution at \( t^n \). This half-time-offset treatment improves the velocity continuity condition at the interface, which can enhance geometric conservation of ALE fluxes for the moving fluid control volumes.

Due to the inherent flexibility of partitioned loosely-coupled schemes, one may construct various iterative coupling procedures at both conceptual and algorithmic levels. However, it may be a challenging task to design a staggered method which provides a good temporal accuracy and long-time stability for coupled fluid–structure interaction problems while maintaining loosely-coupled explicit character. This motivates us to construct a new coupling procedure as described in the next section.

4.3. Combined interface boundary condition method

In the aforementioned staggered schemes, the governing equations of each subdomain have no direct influence along the interface. Instead, the continuity of traction and velocity continuity are enforced sequentially with a time lag. By contrast, in the CIBC method formulation, the interface solution is influenced explicitly by the neighboring subdomains. The CIBC approach achieves improved precision and numerical stability by solving the transformed conditions for the interface quantities based on their space- and time-derivatives [17].

For the coupled FSI problem, the combined residual operators for the Dirichlet condition \( \mathcal{R}^D \) and Neumann condition \( \mathcal{R}^N \) can be constructed as

\[ \mathcal{R}^{[0,N]} \left( \frac{\partial \mathbf{v}_s}{\partial t}, \frac{\partial \mathbf{v}_s}{\partial t}, \frac{\partial \mathbf{P}_s}{\partial n}, \frac{\partial \mathbf{P}_s}{\partial n} \right) = 0 \] along \( \Gamma_i \),

(9)

where \( \rho_f \) denotes the mean density of the fluids, and \( \partial / \partial t \) and \( \partial / \partial n \) denote time derivative and normal spatial derivative of the interface quantities. This construction is inspired by the work of characteristic interface boundary conditions [22] and the variational symplectic integrators [23]. The idea relies on constructing a local discrete energy-preserving property for the staggered stencil between the pair of differential equations. Based on the current solutions on both sides of the interface, these operators calculate successive corrections to the staggered solutions on the interface \( \Gamma_i \). The CIBC scheme requires of a coupling parameter for combining the spatial and temporal derivatives of Neumann and Dirichlet quantities. Next, we present the combined interface conditions which define the aforementioned residual operators \( \mathcal{R}^{[0,N]} \).

4.4. Mathematical formulation

As derived in [9], the conventional interface conditions (1) can be transformed into

\[ \frac{\partial \mathbf{P}_s}{\partial n} = \rho_f \frac{\partial \mathbf{v}_s}{\partial n} \quad \text{on} \quad \Gamma_i, \]

(10)

and

\[ \frac{\partial \mathbf{P}_s}{\partial n} \cdot \mathbf{n}_i = \frac{\partial \mathbf{P}_s}{\partial n} \cdot \mathbf{n}_i \quad \text{on} \quad \Gamma_i, \]

(11)

where (11) is obtained by differentiating the tractions with respect to \( t \). By combining (10) and (11) and using \( \mathbf{n}_i = -\mathbf{n}_s \), we obtain a relation for the velocity of the solid side of the interface:

\[ \rho_f \frac{\partial \mathbf{v}_s}{\partial t} + \omega \frac{\partial \mathbf{P}_s}{\partial n} = \rho_f \frac{\partial \mathbf{v}_s}{\partial n} - \omega \frac{\partial \mathbf{P}_s}{\partial n} \quad \text{on} \quad \Gamma_i, \]

(12)

and, for the traction on the fluid side:

\[ \frac{\partial \mathbf{P}_s}{\partial n} = \rho_f \frac{\partial \mathbf{v}_s}{\partial n} - \omega \frac{\partial \mathbf{P}_s}{\partial n} \quad \text{on} \quad \Gamma_f. \]

(13)

In (12) and (13), \( \omega \) is a dimensional parameter which should be positive and small to ensure that the interface energy is always stable [9]. The parameter \( \omega \) provides the appropriate combination of the applied traction forces with the appropriate corrections into the interfacial acceleration.

4.5. Semi-discretization

The new forms of the interface conditions are constructed for the consistency of the residual interface energy, which is introduced as part of the stability analysis for non-matching meshes in [9]. In (12) and (13), \( \omega \) is a dimensional parameter which should be positive and small to ensure that the interface energy is stable [17]. The parameter \( \omega \) provides the appropriate combination of the applied traction forces with the appropriate corrections into the interfacial acceleration. In the spirit of the Gauss–Seidel iterative procedure, the relationships of (12) and (13) can be written in the explicit (i.e., matrix free) staggered manner between the time levels \( n \) and \( n + 1 \) as

\[ \rho_f \frac{\partial \mathbf{v}_s}{\partial t}^{n+1} = \left( \frac{\partial \mathbf{P}_s}{\partial n} \bigg|_{n} - \omega \left( \frac{\partial \mathbf{P}_s}{\partial n}^{n} - \frac{\partial \mathbf{P}_s}{\partial n}^{n+1} \right) \right) \bigg|_{\Gamma_i}, \]

(14)

for the velocity solution, and

\[ \frac{\partial \mathbf{P}_s}{\partial n}^{n+1} = -\left( \frac{\partial \mathbf{P}_s}{\partial n} \right)^{n+1} \bigg|_{\Gamma_f}, \]

(15)

and for the traction.

Relationships (14) and (15) provide estimates for the traction and velocity corrections between the two successive time levels \( n \) and \( n + 1 \). Instead of applying the velocity and traction conditions directly, we apply a corrected velocity field to the fluid domain \( \Omega_f \)

\[ \mathbf{v}_f^{n+1} = \mathbf{v}_f^n + \delta \mathbf{v}_f^n \quad \text{on} \quad \Gamma_i, \]

(16)

and a corrected traction to the structural domain \( \Omega_s \)

\[ \mathbf{P}_s^{n+1} = \mathbf{P}_s^n + \delta \mathbf{P}_s^{n+1} \quad \text{on} \quad \Gamma_f, \]

(17)

where the velocity correction \( \delta \mathbf{v}_f^n \) is obtained by discretizing (14) as

\[ \delta \mathbf{v}_f^n = \Delta t \rho_f \left[ \left( \frac{\partial \mathbf{P}_s}{\partial n} \right)^n - \omega \left( \frac{\partial \mathbf{P}_s}{\partial n} - \frac{\partial \mathbf{P}_s}{\partial n}^{n+1} \right) \right] \bigg|_{\Gamma_i}, \]

(18)

and the traction correction \( \delta \mathbf{P}_s^{n+1} \) is computed from (15):

\[ \delta \mathbf{P}_s^{n+1} = \Delta t \left[ \left( \frac{\partial \mathbf{P}_s}{\partial n} \right)^{n+1} + \frac{1}{\omega} \left( \rho_f \frac{\partial \mathbf{v}_s}{\partial n}^{n+1} - \frac{\partial \mathbf{P}_s}{\partial n}^{n+1} \right) \right] \bigg|_{\Gamma_f}. \]

(19)
At each time step, the prediction of displacement and the correction of tractions are formed in a sequential manner along the fluid–structure interface as given in Algorithm 3 and shown schematically in Fig. 2b. As derived above, the displacement prediction and the force correction terms are explicit functions of spatial and temporal solutions of fluid and structure along the interface [24]. For the velocity prediction in Algorithm 3, \( \mathcal{R}_{\text{grad}}(P_t, x^f_t) \) denotes traction gradient interface integral, \( \mathcal{R}_{\text{pstate}}(P_t, t) \cdot \mathcal{R}_{\text{pstate}}(P_t, t) \) represent the pressure rate integral along the fluid and structure side interface, respectively, and the interface coupling mass matrix is given by \( M_{i}^{n} = J_{f} \mathbf{N}_{i}^{n} J_{f}^{\top} / \psi \). For the traction correction, \( \mathcal{R}_{\text{adim}}(\rho_s^{n-1} \rho_t^{n+1} \mu; t) \) denotes added mass like integral quantity as an additional term consisting of the product of the structural interface acceleration and the mean fluid density. This term maps the interface acceleration and its associated fluid mass into the load vector, and can be significant when the mass densities of the fluid and structure are about the same order. A detailed discussion about the discretization of these terms can be found in [24].

Algorithm 3. Loosely-coupled explicit discretization of the CIBC method

1. Start from known interface velocity and traction in both subdomains
2. Each FSI cycle between \( t \in [t^n, t^{n+1}] \)
   (a) Solve structure equation for structural velocity prediction
   \[
   M_{f} \frac{d^2 u_{f}^{n+1}}{dt^2} = -\mathcal{R}_{\text{grad}}(P_{f}^{n}, t^{n+1}) \quad \text{in } \Omega_{f} \]
   (b) Solve for velocity prediction
   \[
   \delta v_{f}^{n} = \Delta M_{f} \frac{d}{dt} \left[ \mathcal{R}_{\text{grad}}(P_{f}^{n}, x_{f}^{n}) - \mathcal{R}_{\text{pstate}}(P_{f}^{n}, t) \right] \quad \text{on } \Gamma \]
   (c) Apply Dirichlet kinematic compatibility condition
   \[
   v_{f}^{n+1} = v_{f}^{n+1} + \delta v_{f}^{n+1} \quad \text{on } \Gamma \]
   (d) Set \( x_{f}^{n+1} = x_{f}^{n+1} \) and update fluid dynamic mesh
   \[
   \nabla \cdot \left( (1 + \eta) \nabla \right) x_{f}^{n+1} = 0 \quad \text{in } \Omega_{f} \]
   (e) Advance ALE fluid domain for new conservative variables
   \[
   \frac{d}{dt} \left( \Omega_{f}^{n+1} \mathbf{w}_{f}^{n+1} \right) = -\mathbf{F} \left( \mathbf{w}_{f}^{n+1}, x_{f}^{n+1}, n_{f}^{n+1} \right) \quad \text{in } \Omega_{f} \]
   (f) Solve for traction correction
   \[
   \delta P_{s}^{n+1} = \Delta t \left[ \frac{1}{\mu} \mathcal{R}_{\text{adim}}(\rho_s^{n-1} v_s^{n+1} \mu; t) - \mathcal{R}_{\text{pstate}}(P_{s}^{n+1}, t) \right] - \mathcal{R}_{\text{pstate}}(P_{s}^{n+1}, t) \quad \text{on } \Gamma \]
   (g) Apply Neumann dynamic equilibrium condition
   \[
   P_{s}^{n+1} = P_{s}^{n+1} + \delta P_{s}^{n+1} \quad \text{on } \Gamma \]

In this algorithm, we are considering explicit corrections that are only applied to the right hand side of the fluid and structure equations. This staggered algorithm can also be constructed in a leap-frog manner where the fluid computation is performed at a non-collocated time level \( t^{n+1/2} \) and structural solution at \( t^{n} \). As mentioned earlier, this half-time-offset (or leap-frog treatment) improves the velocity continuity condition at the interface, thereby improving the temporal accuracy and stability [20]. The algorithm can be also extended to a semi-implicit predictor–corrector procedure by forming the structure velocity prediction and the force correction. In [25], the interface force correction terms are formed by iterating between the solutions of fluid and structure via sub-iterations, where the sub-iterations are done by cyclic substitutions at the same time step. The predictor–corrector procedure is based on a nonlinear feedback monitoring, as we go back and forth between the two physical domains.

5. Results of spatial coupling

To evaluate the integrated framework for transient fluid–solid interaction, the accuracy of the load transfer schemes are compared for three test problems involving the coupling of an inviscid fluid with a solid, where the interface is assumed to be impermeable, nonreactive and adiabatic. With these assumptions, the interface can be treated as a contact discontinuity with the following properties: (a) no mass flux; (b) no jump in normal velocity; (c) free-slip boundary condition for the tangential velocity; (d) no jump in the normal stress. All the above conditions and treatments are enforced by the interface coupling schemes through the application of boundary conditions at discrete times.

5.1. 2D elastic piston problem

The first comparative assessment involves the analysis of a transient quasi 1-D FSI problem consisting of expansion fans in the fluid domain coupled with a compression wave in the elastic material domain [8]. This flexible piston problem has a 1-D analytical solution for an inviscid compressible flow and linearly elastic solid. We use a perfect gas (\( \gamma = 1.4 \)) with the physical parameters, an initial density \( \rho_0 = 8.75 \text{ kg/m}^3 \), and an initial pressure \( p_0 = 100 \text{ MPa} \). For the elastic solid, we choose properties similar to that of aluminum, with Poisson’s ratio \( \nu = 0.29 \), density \( \rho_s = 2800 \text{ kg/m}^3 \), and Young’s modulus \( E = 70 \text{ GPA} \). From these properties, we obtain the speed of sound \( c_0 = \sqrt{\rho_0 / \rho_s} = 4000 \text{ m/s} \) in the fluid and the dilatational wave speed \( c_d = \sqrt{E / \rho_s (1 - \nu^2)} = 5224 \text{ m/s} \) in the solid domain for plane stress assumption. We consider the entire coupled system to be quiescent and stress free at \( t = 0 \) for the fluid and solid, respectively. The Eulerian fluid domain is closed by slip walls other than solid interface boundary. The Lagrangian solid domain has roller boundary conditions along the top and bottom walls and traction free condition along the vertical end.

The emphasis of this study is to investigate the effect of non-matching interface discretizations on the evolution of the interface motion. The theoretical solution of this elastic piston problem is fully 1-D, with an interface front moving at a single speed, i.e., without any spatial variation of the interface motion. However, as shown in Fig. 3a, which was obtained for a relatively small mismatch in element size (\( h/I = 0.875 \), with \( h \) and \( I \) respectively denoting the element size along the fluid and solid side of the interface), the N-P and Q-P schemes lead to noticeable spatial variation of the interface velocity profile. In that figure, the local interface velocity in the x-direction \( u_x \) is normalized by the average interface velocity \( u_x^{\text{avg}} \). As also apparent in Fig. 3a, the common refinement scheme leads to an interface that maintains spatial uniformity.

To further assess the impact of the load transfer scheme on the accuracy of the FSI solution, we present on a semi-log scale in Fig. 3b the evolution of the quantity \( \epsilon_p \) which quantifies the deviation of the interface velocity profile from uniformity as
In another comparative study among the three load transfer schemes, we simulated the 2-D fluid–solid interaction problem consisting of a shock in a compressible fluid traveling at a superseismic speed over an elastic half space, i.e., at a speed exceeding the dilatational wave speed of the solid. This problem presents an interesting coupling between fluid and solid solutions since the deformation of the elastic solid behind the traveling shock affects the fluid flow by changing the shock angle and thereby the intensity of the pressure jump across the shock. Furthermore, this problem is also chosen because it has an analytical similarity solution in the frame of the traveling shock and thereby constitutes an excellent test problem for the explicit 2-D FSI framework described in earlier sections and, in particular, the interface transfer schemes.

In this verification study, we select the fluid flow and solid material properties such that the acoustic impedances of the two media are comparable and a significant superseismic coupling occurs. For the linear elastic solid, we use properties similar to those of copper: Poisson’s ratio \( \nu = 0.33 \), density \( \rho_s = 8970 \text{ kg/m}^3 \), and Young’s modulus \( E = 110 \text{ GPa} \). For the fluid, we use a perfect gas with an artificially high initial density, \( \rho_f = 1033 \text{ kg/m}^3 \), and an initial pressure \( p_f = 5.55 \text{ GPa} \). The initial upstream Mach number for this problem is \( M_u = 2.952 \). We consider the unshocked fluid system to be under initial pressure \( \rho_1 \) and solid domain to stress free. The net traction along the interface, \( \mathbf{A}_p \), is computed by subtracting \( \rho_1 \) from the calculated Eulerian fluid pressure field behind the shock \( p_2 \) (see Fig. 4). This yields a zero applied pressure load in the undisturbed fluid region. As indicated in Fig. 4, the boundary conditions for the fluid solver are supersonic inflow on the left side of the domain, and along the mesh boundaries that are not in contact with the fluid, the solid domain has unconstrained and roller boundary conditions.

Fig. 5 presents the spatial variation of the fluid–solid interface deflection \( (u_{ij}) \) obtained with non-matching meshes (symbols), matching meshes (dashed curve) and analytical solution (solid curve). The mesh mismatch for the non-matching mesh case is \( h_f/h_s = 0.875 \). We observe substantial errors in the interface displacement for the traditional load transfer schemes, while the common refinement scheme yields a uniform and smooth interface displacement. This result is further illustrated in Fig. 5b, which presents the relative error in the interface deflection with respect to the reference matching solution. The figure clearly illustrates the oscillatory nature of the solution further behind the shock \( (x < x_{sh}) \) obtained with the two traditional load transfer schemes. The common refinement scheme on the other hand yields a uniform and smooth interface deflection very similar to the matching mesh solution.

### 6. Results of temporal coupling

To assess the accuracy and stability of the staggered schemes, the unsteady ALE fluxes are formed such that the Runge–Kutta method preserve the geometric conservation (see Appendix). The transient structural response is computed using the Galerkin finite element discretization and the Newmark implicit time integration [26]. To investigate the stability and accuracy of the CSS, GSS and CIBC schemes, a set of 1D and 2D FSI problems are simulated in...
Comparison of interface solutions for superseismic shock problem at mesh mismatch ratio $\bar{h}/h_0 = 0.875$: (a) vertical displacement $u_y$, and (b) relative error with respect to the reference matching mesh solution. $x_{th}$ and $V_{th}$ denote the theoretical shock position and speed, respectively.

6.1. 1D elastic piston problem with closed fluid domain

The stability of the CSS, GSS and CIBC coupling methods described in Section 4 is first studied with the aid of the classical, conservative fluid–structure problem of an elastic piston adjacent to a closed fluid domain (Fig. 6). Initially at $t = 0$, the fluid domain has quiescent fluid and the structural domain is stress free. For $t > 0$, the system leads to the oscillating structure when the fluid traction is applied along the interface with corresponding wave propagation in the fluid and solid domains. The amplitudes of interface motion and traction can be obtained for the linearized coupled fluid–structure system. The analytical solution is neutrally stable, i.e., the interface motion and momentum should not grow or decay during the coupled simulations since energy must be conserved. Thus, this problem provides a rigorous testing for analyzing the numerical stability of the staggered methods when the total energy of the system is conserved.

The chosen baseline condition consists in a fluid domain of length $L_f$ filled with a perfect gas ($\gamma = 1.4$) with the physical parameters, an initial density of air $\rho_0 = 1.3$ kg/m$^3$, an initial pressure $p_0 = 1 \times 10^5$ Pa, and the fluid domain size $L_f = 1$ m. To obtain different sets of frequency and mass density ratios, we choose varying values for the structural density $\rho_s$, the Young's modulus $E_s$ and the structural domain size $L_s$. To assess the dependence of the partitioning error on the relative mismatch in physical and geometric properties, two nondimensional parameters are introduced: the relative frequency $f_r = f/(2L_s)$ and the relative mass density $\mu_r = \rho_s/\rho_0$. The structural eigenfrequency is given by $\nu_s = \nu/(2L_s)$ and the fluid acoustic frequency is given by $\nu_f = c_f/(2L_f)$, where $c_f$ and $c_s$ denote the characteristic wave speeds of the fluid and solid, respectively. The parameter $f_r$ characterizes the effects of relative sizes of the subdomains and the interaction between the fluid and the structural waves frequency modes, while the parameter $\mu_r$ defines the relative inertial effects of the fluid–structure coupling. We remark that the mass density ratio determines which physics dominates the coupled dynamics of fluid–structure system. In particular, for $\mu_r \ll 1$ ($\mu_r \gg 1$) the coupled dynamics is dominated by the fluid inertia (structure).

For this particular problem, the fluid and structure solvers are explicit in time with matching time step values ($\Delta t_f = \Delta t_t = \Delta t$) chosen to satisfy the Courant stability condition in both domains ($C_f = C_s \Delta t/\Delta x_0 < 1$ with $k = f$, s). The spatial discretizations are also chosen to match ($\Delta x_f = \Delta x_t$). To quantify the numerical stability of the two staggered methods, the finite size Lyapunov exponent is employed to measure the deviation of phase variables, position and momentum, from their initial amplitudes [27]. The finite size Lyapunov exponent is based on the concept of the average growing time $t_r$, needed for a finite initial error $\delta$ to grow by a factor, $d$, and is defined as

$$\lambda(\delta) = \frac{1}{t_r} \log(d).$$

(21)

Since the energy is conserved within the closed system, the analytical solution for the closed domain problem is neutrally stable and $\lambda(\delta) = 0$. Numerical solutions for which $\lambda(\delta) > 0$ and $< 0$ respectively correspond to unstable and spuriously damped solutions of the coupled system. To avoid potential artifacts near the Courant limits and to resolve the dynamic response, the matching coupling time step is chosen to yield low values of the Courant number in both fluid ($C_f = 0.1$) and structural ($C_s = 0.01$) domains. The results presented herein are generally insensitive for values as high as $C_f = 0.3$ and $C_s = 0.1$.

Fig. 7 shows the influence of relative frequency $f_r$ on the interface solutions of the three staggered methods. Three comments can be made about these results. Firstly, higher values of $f_r$ deteriorate the numerical instability problem in the staggered partitioned methods. This is due to the fact that the errors related to the staggering and partitioning are more prominent when we have a large energy transfer per unit time step. Secondly, the structural prediction and momentum averaging in the GSS method postpones the appearance of numerical stability problem by almost a factor of two compared to the CSS method, but it does not eliminate the
source of instability. Finally, the interface solution for the CIBC method remains stable over the time interval of interest regardless of the value \(f_r\). For the CIBC results, the non-dimensional coupling parameter \(h\) is defined as:

\[
h = \frac{\kappa v_I}{\kappa p_E}, \quad (22)
\]

where the interface velocity can be characterized as \(\|v_I\| = \Delta p / \sqrt{\rho_E}\), with \(\Delta p\) denoting the differential pressure loading. The baseline value of the coupling parameter is taken to be \(\theta = 0.001\) in all 1D studies presented hereafter, for which the interface velocity \(\|v_I\|\) is approximately 1 m/s. Fig. 8 further shows the influence of the relative frequency \(f_r\) on the interface energy transfer (i.e., exchange of work between the fluid and the structure) of the CSS and CIBC staggered methods.

To illustrate the dependence of numerical instability on \(f_r\), Fig. 9 shows the stability indicator \(\lambda\) defined in (21) for a range of frequency ratio. The divergence is measured from the mean oscillatory solution within the normalized time interval \(f_t \in [0, 500]\). In this analysis, we compute the error doubling time \(t_r\), i.e., the time a scheme takes for an initial staggering error \(\delta\) to grow by a factor \(\delta = 2\). The finite size Lyapunov exponent then becomes \(\lambda(\delta) = \log (2)/t_r(\delta)\). From Fig. 9, we observe that \(\lambda\) obtained from the CSS and GSS methods increase nearly monotonically with the frequency ratio \(f_r\). For the same doubling time, the CIBC method has \(\lambda < C25\) with the coupling parameter \(h = 0.001\), in agreement with the theoretical analysis. Small deviations in \(\lambda\) can be attributed to the margin of error in extracting the dynamic response of the interface. The error bar is typically lower than 1%. In the case of low structural frequency (i.e., small \(f_r\)), the staggering-induced energy production in the coupled fluid–structure computation tends to die out due to the action of numerical smoothing required for the stabilization of nonlinear convection terms [28]. On the other hand, high \(f_r\) values yield large staggering errors (i.e., higher values of \(\lambda\)) for the CSS and GSS methods. Fig. 10 shows the dependence of the stability indicator \(\lambda\) on the normalized coupling parameter \(\theta\) used in the CIBC scheme. Energy conserving values of \(\theta\) are in the
range of 0.0005–0.003. Deviation from this range undermines the local energy conservation and eventually yields an unstable behavior similar to that observed in other staggered methods. In addition, the results indicate that it is better to underestimate the coupling parameter $\theta$ for higher values of the relative frequency $fr$. Similar results with regard to the coupling parameter were also obtained for the coupled parabolic thermal equations, for which the CIBC optimal coupling parameter was determined from the Godunov-Ryabenkii’s stability theory and confirmed via numerical experiments [17]. Two additional observations can be derived from Fig. 10. Firstly, decreasing $fr$ increases sharp deviation of $\lambda$ from the stability range of $\omega$ due to the high-sensitivity of staggering with the energy conservation error [10]. Secondly, a half-time-step offset between the fluid and structure computations (solid circles in Fig. 10), alluded to at the end of Section 4, substantially enlarges the lower range of $\omega$ by improving the conservation property of the ALE mesh motion and velocity continuity conditions. This result indicates that improved temporal accuracy enhances the numerical stability of the coupled scheme.

6.2. 1D elastic piston and open fluid domain

We now consider the problem of a semi-infinite fluid domain coupled with a finite elastic media (Fig. 11).
semi-infinite nature of the fluid domain, disturbances associated with the fluid–structure motion of the interface are radiated out of the domain. As a result the energy of the system should be eventually reduced via damped oscillations.

A radiation condition based on the characteristic far-field theory is applied at the left end of the semi-infinite fluid domain [29]. For small motion, an analytical solution for this problem can be developed using the method of characteristics for the linearized Euler equations coupled with an elastic structure. The temporal evolution of interface solution consists of the coupling between the acoustic and the dilatational wave motions [30] in the fluid and structural subdomains, respectively. This problem can be characterized by a single eigenfrequency dictated by the structural motion with \( f_s = c_s/2L_s \).

Fig. 12 depicts the evolution of interface momentum transfer \( P/I \) for the three staggered methods. The relative mass density ratio is typical of aeroelastic coupling: \( \mu = 2000 \). It is apparent that CSS and GSS methods exhibit significant errors with respect to the analytical solution, whose envelope is denoted by the continuous curve and displays the expected decay of the initial perturbation.
In fact, the CSS method starts to build up accumulated errors which eventually cause onset of instability. For the GSS method, the oscillations are qualitatively damped, but the accuracy of the interface evolution is poor in terms of the quantitative damping rate. In particular, the GSS method remains stable without reaching the converged solution predicted by the analytical results. On the other hand, the CIBC method predicts solutions comparable to the analytical envelope. For both lower bound of $h = 0.0005$ and upper bound $h = 0.001$, the interface momentum transfer decays to the analytical convergence point. However, the evolution of the momentum transfer for $h = 0.001$ is slightly more diffusive than $h = 0.0005$ compared to the analytical envelope at the cost of an additional stabilization.

6.3. 2D subsonic flow-shell problem

To further illustrate the effect of the two staggered methods on the numerical stability of FSI simulations in the case of non-matching interface discretizations, we now study the 2D problem of a subsonic duct flow over a curved thin-shell structure (Fig. 13). Since transient fluid–structure interaction problems may possess a wide variety of self-excited vibrations and physical instabilities, we focus here on situations where the coupled system is physically stable, i.e., for which the coupled solution does not grow indefinitely in time. To prevent such physical instabilities, test conditions are chosen with: (a) a subsonic flow speed ($M_1 < 0.18$) much lower than the critical divergence speed [31]; and (b) a magnitude of unsteady fluid loading much lower than the in-plane elastic buckling load [32]. The load level and shell geometries are also chosen such that the linear kinematic description of the shell response is appropriate.

For the thin shell, we use linear elastic properties similar to those of the aluminum: Poisson’s ratio $\nu_s = 0.30$, mass density $\rho_s = 2600 \text{ kg/m}^3$, and Young’s modulus $E_s = 70 \text{ GPa}$. The length $L_s$, the thickness $h_s$, and the included angle $\chi_s$ of the thin-shell are variables to chosen to achieve different values for the structural frequency $f_s$ and are chosen in the range: $0.3 \text{ m} \leq L_s \leq 1.0 \text{ m}$, $0.002 \text{ m} \leq h_s \leq 0.006 \text{ m}$, and $12^\circ \leq \chi_s \leq 45^\circ$. For the fluid, a perfect gas ($\gamma = 1.4$) is assumed with the freestream pressure $p_\infty = 0.3039 \text{ MPa}$, Mach number $M_\infty = 0.18$ and mass density $\rho_\infty = 1.3 \text{ kg/m}^3$. The size of fluid domain is assumed infinitely long, and a truncation is applied at the discrete level with the far-field non-reflective properties of the inflow and outflow boundary conditions [33]. The height of fluid domain is chosen as $H_f = 1.25 \text{ m}$. For this problem, we model the fluid flow with the compressible Euler equations by assuming that the viscous time scales far exceed the convective and acoustic time scales. Therefore, this problem is characterized by three fundamental frequencies corresponding to the uncoupled characteristic time scales: (a) the shell free-vibration frequency, $f_s = (B/L_s^2)\sqrt{E_s I_s/\rho_s A_s}$, where $B$ is a non-dimensional frequency parameter [34], $I_s$ is the moment of inertia, and $A_s$ is the cross-sectional area; (b) the fluid acoustic frequency, $f_a = c/2H_f$ associated with the time to travel of acoustic wave across the width of the duct; (c) the fluid convective frequency $f_c = (M_\infty c)/L_s$, related to the freestream travel speed $U_\infty = M_\infty c$ and the length of shell $L_s$. The frequency ratio for this problem is defined as $f_r = f_c/f_s$.

![Fig. 13](image-url) (a) Schematic of subsonic flow past over a thin shell structure. (b) Partial view of the fluid–structure mesh with non-matching nodes along the interface.
The fluid grid is unstructured with the gradient-based adaptive clustering near the fluid–structure boundary to reduce the total number of fluid elements. The explicit fourth-order Runge–Kutta scheme and the second-order accurate finite-volume discretization are employed in the fluid domain. For the structural domain, we use the implicit Newmark scheme for time integration with the parameters corresponding to the energy-conserving solutions \([26]\) \((\beta = 1/4 \text{ and } \gamma = 1/2)\). To capture the structural bending response accurately, we use quadrilateral elements with the mixed-enhanced finite element formulation \([35]\) to discretize the thin-shell (Fig. 13b). These elements provide a high-coarse mesh accuracy with a large aspect ratio in the bending dominated problems. This combination of discretizations clearly reveals the need and advantage of the nonmatching meshes along the interface (Fig. 13b). To evaluate interface integrals accurately and conservatively, we employ the common-refinement based scheme across nonmatching meshes \([9]\). Prior to solving the coupled problem, standalone fluid and structural solutions are verified with the benchmark solutions for the classical Ni’s test case of a subsonic flow over the 10% thick circular bump \([36]\), and the free vibration results of the thin shell \([34,37]\). However, to the best of our knowledge, this coupled fluid–structure problem does not have benchmark numerical or analytical solutions in the literature when a nonlinear compressible flow interacts with a finite elastic shell. However, from an eigenmode analysis of subsonic far-field boundary conditions \([29]\), it is expected that the structural transient effects decay exponentially. In \([29]\), it was shown that the decay rate or aerodynamic damping of small flow perturbations depends on the free stream Mach number \(M_s\). For the coupled problem of interest, a similar long-time steady behavior is expected from the staggered methods, as the elastic energy of the shell is balanced with the acoustic energy radiated during the period of oscillation.

The focus of this study is thus to understand the possible scenarios in which the staggering errors may affect the coupled long-time steady behavior for the certain frequency ratios. For this application, as predicted by the 1D analysis, we directly utilize the CIBC treatment with the half-time offset stencil and a coupling parameter \(\sigma = 0.00075 \text{ (s/m)}\) from the optimal interval estimate. To simulate this coupled problem, the structure is first held fixed and the steady-state flow around the thin shell is computed. A nonlinear pressure distribution develops over the structure. The flexible structure is then released, except at its clamped ends, and starts vibrating under the pressure of the flow, inducing complex interactions between the flow and the thin structure. Fig. 14 shows a typical evolution of the mid-point vertical displacement \(u_{\text{mid}}\) along the interface for the CIBC method at the frequency ratio \(f_s = 0.5\) with matching time steps \(\Delta t_f = 1.0 \times 10^{-3} \text{ s}\). A Fourier analysis of the aeroelastic oscillations indicates that there is a superimposition of two frequencies: those of the structural excitation \(f_s\) and those of the acoustic reflections \(f_a\).

To investigate the impact of the staggered scheme on the stability of FSI simulations, we compare in Fig. 15 the results obtained of the CSS, GSS and CIBC schemes for the frequency ratio \(f_s = 2.5\) (which corresponds to 520 Hz) and for two values of the substepping factor \(\eta_s = \Delta t_s/\Delta t_f = 1\) (i.e., matching time steps) and \(\eta_s = 5\). Fig. 15a presents the evolution of the mid-shell deflection obtained with matching time steps \((\Delta t_s = \Delta t_f = 1.0 \times 10^{-5} \text{ s})\), clearly pointing to instability inherent in the CSS staggering scheme, while the GSS and CIBC solutions lead to a slow decay of the perturbation associated with the initial stage of the coupled simulation. The GSS method, which is based on a higher-order structural prediction, eventually asymptotes to a neutrally stable interface motion. This nonphysical behavior is not present in the CIBC solution, which properly decays towards the coupled steady state response of the fluid–structure problem. This trend is even more pronounced for the case of \(\eta_s = 5\) shown in Fig. 15b. While the CIBC scheme yields slowly decaying results similar to those shown in Fig. 15a, the instability of the CSS and GSS schemes are further emphasized, leading to a diverging solution.

These observations are summarized in Fig. 15c, which shows the dependence of the computed decay factor on the relative frequency factor \(f_s\), as computed by the CIBC and CSS staggering schemes. The open symbols correspond to the matching time step cases, while the filled symbols shown for \(f_s = 2.5\) denote the results obtained for a subcycling ratio of 5. As remarked earlier, the coupled transient solutions should be of a converging nature due to a continuous loss of the elastic energy through the acoustic radiations. While the coupled damping rate is not analytically tractable, the decay factor \(\lambda_{\infty}\) can be estimated by assuming a rigid shell (i.e., no structural effects) from the eigenmode analysis \([29]\) of the far-field boundary conditions: \(\lambda_{\infty} \approx -\frac{1}{1 + M_s} \left[ \log \sqrt{\frac{1 + M_s}{1 - M_s}} \right] \) by assuming the linearized Euler flow and non-reflective boundary conditions. As shown in Fig. 15c, we expect the coupled steady response to approach the decay rate \(\lambda_{\infty}\) for a very fast vibrating structure in a duct flow. For the CIBC method, the decay rate is negative and shows its effectiveness in eliminating structural transient behavior. However, the CSS and GSS methods exhibit non-physical divergence, especially for the substepping (subcycling) factor \(\eta_s = 5\).

As apparent from the results presented in this section, the results of the simple 1D analysis summarized in the previous section extend directly to more complex 2D fluid–structure simulations involving curved interfaces and unstructured meshes. This finding is consistent with the previous work on the precision and conservation of load transfer across nonmatching meshes \([8,9]\). It is generally true that instabilities first occur with a purely 1D eigenmode with a spatial variation in the direction normal to the interface but no variation along the interface. Therefore, these results of 1D and 2D simulations are expected to be applicable to more complex situations in which the 3D equation of motion is used to structures and the 3D Navier–Stokes equations are used to model the behavior of the fluid.

Finally, the computational costs are an important factor when selecting a numerical scheme for large-scale simulations. As discussed in \([8]\), the size of the interface discretization is typically orders of magnitude smaller than the fluid and solid meshes. Therefore, the cost of additional interface variables typically represents a small fraction of the overall cost.

7. Conclusions

This article has focused on the impact of load transfer and staggering schemes for spatial and temporal coupling on the accuracy
and stability of transient FSI simulations. For the spatial coupling, we have systematically assessed three different conservative load transfer schemes, namely, node-projection, quadrature-projection, and common-refinement based schemes for nonmatching meshes. In the traditional node- and quadrature-projection schemes, the integrand exhibits discontinuities that generally lead to load transfer errors depending on the fluid–solid mesh mismatch along the interface. By utilizing a common refinement and integrating over the subintervals, the error becomes independent of the mismatching along the interface, such that solution with nonmatching meshes was effectively identical to the solution of the matching meshes. These findings were demonstrated with the aid of two transient fluid–solid interaction problems.

With regard to the temporal coupling, we have described and assessed a new staggering scheme, referred to as CIBC, that incorporates more closely the physical models on either side of the interface. By constructing the combined conditions from the interface continuity conditions, we are able to damp out the energy residual produced at the interface and can achieve accuracy and stability of the coupled system as high as the individual fluid and structure solvers. The amount of correction is controlled by a coupling parameter, which is a dimensional positive metric required to combine spatial and temporal derivatives. The optimal value of the non-dimensional coupling parameter, $\theta$, seems to be approximately 0.0005–0.003. In addition to stability aspects, the issue of temporal accuracy of the loosely-coupled procedure for the fluid–structure system has been addressed. By comparing with the analytical solutions, we have verified that the CIBC scheme is not only more stable than the CSS and GSS methods, but also more accurate. These results of numerical stability are translated to complex 2D fluid–structure problem using the coupling parameter computed from the 1D analysis. The GSS method has provided significantly improved stability and accuracy over the CSS method for the open elastic piston and flow-shell problems. However, the GSS method can lead to divergence for high frequency ratio for the closed piston problem and for large sub-cycling factor in the case of the 2D flow-shell problem. The CIBC scheme does not exhibit divergence behavior for the high frequency ratio and the large sub-cycling factor.

Future work is needed to confirm that the CIBC scheme with common refinement methods work with 3D FSI system and with unsteadiness due to turbulence. In addition, a strong stability condition and a closed-form expression for the coupling parameter should be formulated for a nonlinear FSI system.

Acknowledgments

The authors wish to acknowledge support from the Center for the Simulation of Advanced Rockets (CSAR) funded by the US Department of Energy under subcontract B523819 and from the Computational Science and Engineering (CSE) program at University of Illinois.
Appendix A. Analytical proof for geometrical conservation law

For the geometric conservation law [38], the basic requirement is that any ALE computational method should be able to predict exactly trivial solution of a uniform flow:

\[
\int_{\Omega}^{t} \frac{d\Omega}{dt} - \int_{\Omega}^{t} \frac{d\Omega}{dt} \int_{\Omega(t)} \nabla \cdot \mathbf{x} d\Omega dt = \int_{\Gamma}^{t_{1}} \int_{\Gamma(t)} \left( \mathbf{F} \cdot \mathbf{n} \right) dA dt,
\]

which states that the change in volume (or area, in 2-D) of each element from \( t^n \) to \( t^{n+1} \) must be equal to the volume (or area) swept by the element boundary during the time interval for the given fluid mesh velocities \( \mathbf{v} = (\mathbf{x}_t - \mathbf{x})/\Delta t \). If the fluid grid positions \( \mathbf{x} \) and velocities \( \mathbf{v} \) are explicitly known functions of time, then the ALE fluid system can be written as

\[
\int_{\Gamma}^{t_{1}} \left( \nabla (\mathbf{w}^{(1)}) (\mathbf{w}^{(1)})^\top - \nabla \mathbf{v} \cdot \mathbf{n} \right) dA = 0 \quad \text{(A.2)}
\]

where \( \mathbf{w} \) and \( \mathbf{n} \) denote the temporal evaluation stage for the RK scheme and for the mid-point rule \( k = 1/2 \). By using properties of the Jacobian determinant:

\[
\det (I + \Delta t \mathbf{A}^t) = 1 + \Delta t \det (\mathbf{A}) + \Delta t \text{trace}(\mathbf{A}) + \Delta t 
\]

and geometric mapping relations by the chain rule of differentiation at the \( \eta^m \) mesh configuration:

\[
\frac{\partial (x^1)}{\partial \eta^m} = \frac{\partial (x^1)}{\partial x^1} \left( x^{n+1} + \frac{\Delta t}{2} \nabla \mathbf{x} \right), \quad \frac{\partial (x^n)}{\partial x^n} = \frac{\partial (x^n)}{\partial \eta^n} \left( x^{n+1} - \frac{\Delta t}{2} \nabla \mathbf{x} \right),
\]

we obtain

\[
\sum_{n=1}^{N} \left[ \det (I + \Delta t \frac{\Delta t}{2} \mathbf{A}) - \det (I - \Delta t \frac{\Delta t}{2} \mathbf{A}) \right] = \Delta t \sum_{n=1}^{N} \mathbf{a}_n \nabla \cdot \mathbf{x}_n
\]

where

\[
\text{Det}(I + \Delta t \frac{\Delta t}{2} \mathbf{A}) = 1 + \Delta t \left( \frac{\Delta t}{2} \mathbf{A} \right) + \frac{\Delta t}{2} \mathbf{A}, \quad \text{Det}(I - \Delta t \frac{\Delta t}{2} \mathbf{A}) = 1 - \Delta t \left( \frac{\Delta t}{2} \mathbf{A} \right) - \frac{\Delta t}{2} \mathbf{A}, \quad \text{Det}(I + \Delta t \mathbf{A}) = 1 + \Delta t \mathbf{A}
\]

and \( \mathbf{E}(\mathbf{A}) \equiv 0 \) in 2-D, and \( \mathbf{E}(\mathbf{A}) \equiv \mathbf{E}(\mathbf{A}) \) in 3-D.

For the mid-point rule, which is a special case of the RK scheme, \( \Delta t = 1 \) and \( \mathbf{a}_0 = 1 \), and the left hand side of (A.5) yields the right hand side in the exact arithmetic. Similarly, from the consistency of the RK scheme

\[
\sum_{j=1}^{k} \mathbf{a}_j = \mathbf{c}_k
\]

we can satisfy (A.1), which states that the change in volume (or area, in 2D) from \( t^n \) to \( t^{n+1} \) must be equal to the volume (or area) swept by an individual control volume boundary during that time interval. Since the \( \mathbf{O}_j \) on the left-hand side of (A.9) can be computed exactly as functions of the grid positions, this amounts to requiring the exact computation of the unknown values of \( \mathbf{u}_j \) at the \( j \)th RK stage. For the four-stage explicit RK scheme, \( \mathbf{u}_j \) unknowns can thus be obtained exactly by solving the linear system by forward substitution [39].

References


