


RESEARCH ARTICLE

Open Access



One-way coupled fluid–beam interaction: capturing the effect of embedded slender bodies on global fluid flow and vice versa

Nora Hagemeyer^{1*} , Matthias Mayr^{1,2}, Ivo Steinbrecher¹ and Alexander Popp¹

*Correspondence:
nora.hagemeyer@unibw.de

¹Institute for Mathematics and Computer-Based Simulation, University of the Bundeswehr Munich, Neubiberg, Germany
Full list of author information is available at the end of the article

Abstract

This work addresses research questions arising from the application of geometrically exact beam theory in the context of fluid-structure interaction (FSI). Geometrically exact beam theory has proven to be a computationally efficient way to model the behavior of slender structures while leading to rather well-posed problem descriptions. In particular, we propose a mixed-dimensional embedded finite element approach for the coupling of one-dimensional geometrically exact beam equations to a three-dimensional background fluid mesh, referred to as fluid–beam interaction (FBI) in analogy to the well-established notion of FSI. Here, the fluid is described by the incompressible isothermal Navier–Stokes equations for Newtonian fluids. In particular, we present algorithmic aspects regarding the solution of the resulting one-way coupling schemes and, through selected numerical examples, analyze their spatial convergence behavior as well as their suitability not only as stand-alone methods but also for an extension to a full two-way coupling scheme.

Keywords: Fluid–structure interaction, Nonlinear beam theory, Finite element method, Immersed boundary method, Mixed-dimensional modeling, 1D–3D coupling

Introduction

The interaction of rod-like structures with fluid flow plays an important role in a broad spectrum of applications varying from biomechanical to industrial processes. Examples include the interaction of endovascular devices with blood flow [1], coiling during the treatment of cerebral aneurysms [2], the effect of submerged vegetation on surrounding fluid [3,4], the application of brush seals as well as the use of fibrous coverings in flow control [5,6], to name a few. The numerical modeling of problems involving such rod-like structures with classical continuum-based finite elements poses a challenge because it promptly leads to locking effects as well as very large system sizes. In order to ensure a well-posed formulation as well as an efficient usage of computational resources, applications can often benefit from employing one-dimensional (1D) beam theory to model the slender bodies compared to a full three-dimensional (3D) solid formulation.

The application of dimensionally reduced models to adequately describe complex behavior of slender bodies has a long-standing history. The formulations in this paper greatly build on the extension of such models to the nonlinear regime of finite deformations in three dimensions. Important to mention in this context is the extension to a finite strain problem in 3D by Simo [7] based on the work by Reissner [8]. The formulations proposed in these works fall in the group of geometrically exact beam theories and constitute the basis of computationally efficient and rather well-posed problems for systems of slender bodies. The actual beam implementations used in the following constitute an extension and application of the above ideas by Meier et al. [9–11].

These reduced-dimensional formulations can be used to model highly complex systems of slender struts. Tambača et al. [12] propose a general one-dimensional vascular stent model, which is utilized in [13] to efficiently simulate the behavior of coronary stents under physiologically reasonable conditions. As an extension of that, the behavior of this one-dimensional stent model is compared to the results of a fully resolved three-dimensional simulation in [14]. A difference of less than 6% in the displacement magnitude at a complexity reduction, in terms of degrees of freedom (DoFs), of more than 400 was observed.

Staying with the application of stenting procedures for the moment, it becomes clear that not only the behavior of the stent but especially its interaction with the arterial wall, be it chemical in the case of drug-eluting stents, or mechanical in nature, is of interest. Additionally, “[a]n extension of [such a] model to account for flow perturbations would be beneficial and is in order” [14, p. 614], in particular to model growth and restenosis as a response to oscillating wall shear stresses (WSS). In general, this necessitates approaches for the interaction between the 1D beam model with classical continuum-based equations describing the blood flow and the mechanics of the vessel wall.

Such mixed-dimensional models are not restricted to endovascular devices, but naturally arise in a multitude of applications including structures with high slenderness ratios. The solution of these mixed-dimensional interaction problems leads to new challenges compared to equal-dimensional problems. The most prominent among them is the question of how to couple three-dimensional variables with 1D stress resultants as well as the challenge of transferring values between unavoidably geometrically non-matching meshes.

In the area of fluid-structure interaction (FSI), the immersed boundary method (IBM) [15–17] constitutes a well-studied method for mixed-dimensional interactions. Nearly five decades ago, Peskin first explored the idea of embedding a one-dimensional body representing the response of a thin structure within a two-dimensional fluid domain in order to simulate the blood flow in the vicinity of a heart valve [15]. At the time, Peskin modeled the blood flow via the incompressible Navier–Stokes equations spatially discretized with a finite difference method (FD). The force exerted by the heart valve itself on the fluid around it was modeled as a direct linear response to the fluid velocity. The interpolation of the fluid velocity and force response between the two meshes was realized by using so-called delta functions as numerical approximations of Dirac functions.

Since that time a significant amount of research was conducted on the extension and application of the IBM to fully-resolved equal-dimensional 2D and 3D FSI problems based on fictitious domain methods, cf. [1, 18–21] among many others. In such methods, the domain occupied by the immersed structure is filled by a fictitious fluid volume,

which usually leads to a coupling of the involved FSI values on the fictitious domain volume and/or its surface. An exception represents the work of Baaijens in [20], in which applications of his proposed fictitious domain/mortar finite element methods to slender bodies are shown. The slender bodies are modeled with continuum-based equations, while the coupling interface is reduced to only one side of the fictitious domain. Comparisons with fully coupled Arbitrary Lagrangian Eulerian (ALE) based methods show the validity of this simplification.

In contrast, one-dimensional beam formulations are embedded in a two-dimensional fluid in [4, 22]. In [22], the coupling of the fluid and the beam is realized with moving composite meshes interacting on the beam surface, while an IBM-type coupling of the beam surface with the background mesh is presented in [4]. Finally, Huang et al. [23] present results for an IBM-type method coupling a one-dimensional beam formulation with a three-dimensional fluid. Again, the beam is coupled with the Navier–Stokes equations on the reconstructed beam surface, making all of the mentioned approaches surface-to-volume coupling schemes as categorized in [24]. Within these approaches, the slender body is modeled using one-dimensional equations, the fluid domain consists of the entire simulation domain, and the fluid–beam interaction quantities are coupled on the reconstructed beam surface. This makes it necessary to reconstruct the beam surfaces to be coupled with the fluid equations, and in turn, to exchange relevant data between the beam surface and its centerline. As a result, the fluid domain has to be massively refined in order to resolve the surface mesh tying, and also the coupling procedure itself rapidly grows more complex in terms of computational efficiency, somewhat reducing the computational advantages of employing a reduced dimensional beam model. The work of Tschisgale et al. [25] represents an intermediate step, since the beam surface does not have to be reconstructed. Instead, classical regularized delta functions, depending on the beam radius, are used to add fluid–beam interaction forces to the overall problem. This method, nevertheless, also necessitates the use of finely resolved background meshes in order for the delta functions to have a width of multiple fluid elements. A truly mixed-dimensional fluid–beam interaction method, that allows for relatively coarse background meshes, was discussed only in [26]. Here, the coupling was realized on the beam centerline and applied to the simulation of a transcatheter heart valve. The computational results of the biomechanical problem look promising, however, no numerical study of the necessary assumptions for the validity of the method nor its limitations were presented.

Within this contribution, we now propose an immersed mixed-dimensional FSI formulation, embedding slender bodies modeled by geometrically exact beam theory within a three-dimensional incompressible fluid flow. Both, the beam formulation and the employed incompressible isothermal Navier–Stokes equations, are discretized with the finite element method (FEM). The proposed formulation is based on the assumption that the beam diameter is smaller than the characteristic fluid element diameter, and that the development of global flow phenomena, in contrast to interface phenomena, is of primary interest. With such application cases in mind, we will follow the simplification ideas of Baaijens [20, 26] and directly couple the one-dimensional beam centerline to the three-dimensional continuum equations following a Gauss-point-to-segment (GPTS) type approach, e.g. as commonly used in contact mechanics. This work constitutes the first rigorous numerical study of a truly mixed-dimensional 1D–3D coupling approach to capture global effects of the interaction of nonlinear beam elements with 3D fluid flow in

a manner that enables the use of a relatively coarse background mesh compared to the diameter of the slender beam. Our method, thus, allows for a very efficient solution of the FBI problem in terms of the required number of unknowns. For highly slender embedded fibers, the use of a relatively coarse fluid mesh is crucial in order to conserve benefits of using a reduced-dimensional structure formulation in terms of the system size, solver time as well as the evaluation of the coupling terms itself. These advantages become even more pronounced for applications including a large number of embedded fibers. Similar coupling approaches have already shown promising results for solid-beam interaction in our previous work [24] and for the modeling of tumor growth in [27].

The remaining part of this paper will be structured as follows: In “[Problem formulation](#)”, we will present the single field equations as well as the continuous FBI problem. “[Discretization and numerical integration](#)” will briefly outline the discretization methods used to treat the partial differential equations (PDEs), the load and motion transfer schemes used to interpolate between the two domains as well as the numerical integration of the coupling terms. In “[One-way coupling algorithms](#)”, we will introduce one-way coupling algorithms used for the subsequent numerical results. In “[Numerical examples](#)”, we will proceed to demonstrate the applicability of the proposed approach by validating results of the one-way coupling schemes in the two special cases of (i) very light fibers as well as (ii) almost rigid beam structures, by discussing numerical features of the proposed approach as well as studying its behavior under uniform mesh refinement with respect to a 3D reference solution.

Problem formulation

In this section we briefly recount the governing equations for the fluid field, namely the incompressible isothermal Navier–Stokes equations for Newtonian fluids, as well as the geometrically exact beam formulations applied to the numerical examples in “[Numerical examples](#)”. Both fields are coupled via an IBM-type approach applied on the beam centerline in the current configuration as detailed in “[Mixed-dimensional fluid–beam interaction](#)”. For improved readability, the superscript $(\cdot)^f$ shall denote fluid-related quantities. For the beam equations, we will use the abbreviation $(\dot{x}) := \frac{\partial x}{\partial t}$ for the time derivative of x , and $(x') := \frac{\partial x}{\partial s}$ for the derivative with respect to the curve parameter s as is common practice in beam theory. In turn, we will abstain from a rigorous use of a superscript to denote all beam-related quantities. Nevertheless, the superscript $(\cdot)^b$ will be used to highlight beam-related quantities where appropriate.

Geometrically exact beam theory

We will use geometrically exact beam models in the sense that “the relationships between the configuration and the strain measures are consistent with the virtual work principle and the equilibrium equations at a deformed state regardless of the magnitude of displacements, rotations and strain” [28, p. 1126]. The two geometrically exact beam formulations treated within this work fall into the category of semi-induced beam theories, meaning that the kinematic equations are consistently derived from the three-dimensional continuum theory while the constitutive equations are directly postulated on the one-dimensional geometry. In contrast to fully induced methods this approach ensures the fulfillment of important mechanical principles such as the equilibrium of forces [9].

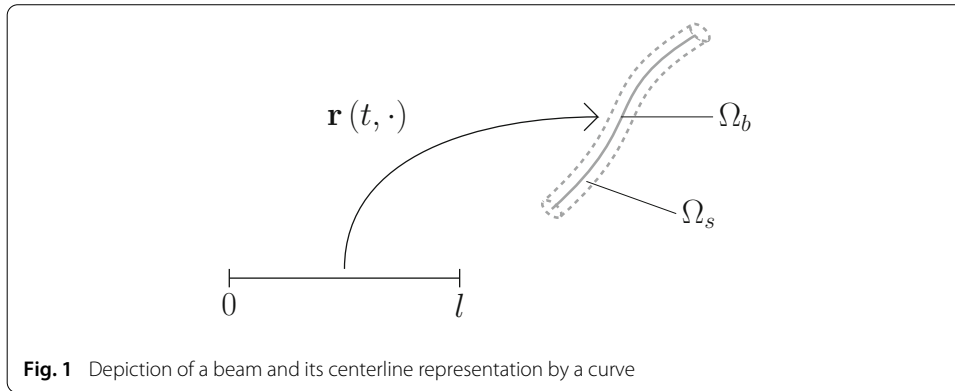


Fig. 1 Depiction of a beam and its centerline representation by a curve

In order to write the two particular beam formulations in the upcoming subsections, namely the Simo–Reissner (SR) model and a torsionfree (TF) variant, in a compact form, let

$$\mathcal{V}_B^p := \left\{ v : [0, l] \rightarrow \mathbb{R}^3 : \left\| \frac{\partial^\alpha v}{\partial s} \right\|_B < \infty, \alpha \leq p \right\} \quad \text{for } p \in \{1, 2\}, \tag{1}$$

be two different spaces of curves on $[0, l]$ with additional requirements on the smoothness of higher order derivatives of the contained curves. Both spaces shall be endowed with the inner product $(\cdot, \cdot)_B := \int_{\mathbb{R}} (\cdot, \cdot)_{\mathbb{R}^3} ds$ and the associated norm $\|\cdot\|_B := \sqrt{(\cdot, \cdot)_B}$. Here, $(\cdot, \cdot)_{\mathbb{R}^3}$ simply denotes the scalar product of two vectors in 3D and $\|\cdot\|_{\mathbb{R}^3} := \sqrt{(\cdot, \cdot)_{\mathbb{R}^3}}$ its associated norm. Furthermore, in the upcoming subsections, we will denote the cross product of two vectors \mathbf{a} and \mathbf{b} using the cross product operator $\mathcal{S}(\mathbf{a})\mathbf{b} := \mathbf{a} \times \mathbf{b}$.

In general, the state of a beam at each time $t \in I := [0, T]$ can be characterized by the position of its centerline $\mathbf{r}(t, \cdot) \in \mathcal{V}_B^p := \mathcal{V}_B^p$, for $p \in \{1, 2\}$, and the cross-section rotation vector $\boldsymbol{\psi}(t, \cdot) \in \mathcal{V}_\theta := \mathcal{V}_B^1$. With the help of Rodrigues’ rotation formula, the rotation vector can be used to define a rotation matrix $\mathbf{\Lambda} = \mathbf{\Lambda}(\boldsymbol{\psi}(t, s)) \in SO(3)$ in the special orthogonal group $SO(3)$ for all $s \in [0, l], t \in I$. Fig. 1 visualizes the image of the centerline curve $\Omega_b := \Omega_b(t) := \mathbf{r}(t, [0, l]) \subset \mathbb{R}^3$, as well as the 3-dimensional domain $\Omega_s := \Omega_s(t) \subset \mathbb{R}^3$ denoting the beam’s current configuration, at a specific time t .

To maintain a compact notation where possible, we introduce the solution state $\boldsymbol{\eta}^b \in \mathcal{V}_B$ summarizing all relevant unknowns, such as the centerline position \mathbf{r} and the rotation vector $\boldsymbol{\psi}$, to represent the state of the regarded beam. The exact form of $\boldsymbol{\eta}^b$ and \mathcal{V}_B depends on basic modeling assumptions and therefore differs for the two beam models, which will be introduced in the upcoming subsections.

For now, using the presented general notation, we can complete the general beam problem by introducing the Dirichlet boundary conditions $\boldsymbol{\eta}_D$ and an initial state $\boldsymbol{\eta}_0$. Then, the state of the beam can be characterized as solution $\boldsymbol{\eta}^b$ of the following problem:

Problem 1 Find $\boldsymbol{\eta}^b \in L^2(I, \mathcal{V}_B + \boldsymbol{\eta}_D)$ such that

$$\int_0^T \mathbf{a}^b(\boldsymbol{\eta}^b; \delta\boldsymbol{\eta}^b) - \mathbf{b}^b(\delta\boldsymbol{\eta}^b) dt = 0 \quad \forall \delta\boldsymbol{\eta}^b \in \mathcal{V}_B, \tag{2}$$

with $\boldsymbol{\eta}^b = \boldsymbol{\eta}_0$ a. e. for $t = 0$, and $\|\mathbf{r}'(0, \cdot)\|_{\mathbb{R}^3} = 1$ a. e. on $[0, l]$.

The exact form of the semi-linear form $\mathbf{a}^b(\boldsymbol{\eta}^b; \delta\boldsymbol{\eta}^b)$, the linear form $\mathbf{b}^b(\delta\boldsymbol{\eta}^b)$, and the corresponding solution space \mathcal{V}_B will be given in the upcoming subsections for two particular beam models.

Remark 2 In problem 1, the additional condition $\|\mathbf{r}'(0, \cdot)\|_{\mathbb{R}^3} = 1$ means that $\mathbf{r}(0, \cdot)$ represents an arc-length parameterized curve on $[0, l]$, or, in other words, l defines the initial length of the beam.

Simo–Reissner beam model

The Simo–Reissner (SR) model is based on the assumptions of plane, rigid cross-sections, but does not introduce any additional kinematic constraints on the beam. For further details on the derivation, see e.g. [7].

For the density ρ^b , shear and Young’s modulus G and E , respectively, the cross-section area A , the reduced cross-section values A_1 and A_2 , the torsional moment of inertia I_T , and the principal moments of inertia I_2, I_3 , the constitutive matrices $\mathbf{C}_M, \mathbf{C}_F$, and the inertia tensor \mathbf{C}_ρ are given as follows:

$$\mathbf{C}_M := \begin{pmatrix} GI_T & 0 & 0 \\ 0 & EI_2 & 0 \\ 0 & 0 & EI_3 \end{pmatrix}, \mathbf{C}_F := \begin{pmatrix} EA & 0 & 0 \\ 0 & GA_2 & 0 \\ 0 & 0 & GA_3 \end{pmatrix}, \tag{3}$$

$$\text{and } \mathbf{C}_\rho := \begin{pmatrix} \rho^b(I_2 + I_3) & 0 & 0 \\ 0 & \rho^b I_2 & 0 \\ 0 & 0 & \rho^b I_3 \end{pmatrix}.$$

With the deformation measures $\boldsymbol{\Gamma} := \boldsymbol{\Lambda}^T \mathbf{r}' - \mathbf{e}_1$, and the solution state $\boldsymbol{\eta}^b := (\mathbf{r}, \boldsymbol{\psi})$, the semi-linear form in Problem 1 is defined as

$$\begin{aligned} \mathbf{a}^b(\boldsymbol{\eta}^b; \delta\boldsymbol{\eta}^b) := & (\rho^b \mathbf{A} \ddot{\mathbf{r}}, \delta \mathbf{r})_B + (\boldsymbol{\Lambda} \mathbf{C}_F \boldsymbol{\Gamma}, \delta \mathbf{r}')_B \\ & + (\boldsymbol{\Lambda} \mathbf{C}_M \boldsymbol{\Omega}, \delta \boldsymbol{\theta}')_B - (\boldsymbol{\Lambda} \mathbf{C}_F \boldsymbol{\Gamma}, \mathbf{S}(\delta \boldsymbol{\theta}) \mathbf{r}')_B \\ & + (\mathbf{S}(\mathbf{w}) \mathbf{c}_\rho \mathbf{w} + \mathbf{c}_\rho \dot{\mathbf{w}}, \delta \boldsymbol{\theta})_B, \end{aligned} \tag{4}$$

and the linear form defaults to

$$\mathbf{b}^b(\delta\boldsymbol{\eta}^b) := (\mathbf{f}, \delta \mathbf{r})_B + [\tilde{\mathbf{f}}^T \delta \mathbf{r}]_0^l + (\mathbf{m}, \delta \boldsymbol{\theta})_B + [\tilde{\mathbf{m}}^T \delta \boldsymbol{\theta}]_0^l. \tag{5}$$

Here, we use the external forces \mathbf{f} , the external moments \mathbf{m} , the point forces $\tilde{\mathbf{f}}$, the point moments $\tilde{\mathbf{m}}$, the angular velocity vector \mathbf{w} such that $\mathbf{S}(\mathbf{w}) = \dot{\boldsymbol{\Lambda}} \boldsymbol{\Lambda}^T$, and the material curvature vector $\boldsymbol{\Omega}$ such that $\mathbf{S}(\boldsymbol{\Omega}) \mathbf{a} = \boldsymbol{\Lambda}^T \boldsymbol{\Lambda}' \mathbf{a}$ for all $\mathbf{a} \in \mathbb{R}^3$.

For the above integrals to be well defined, the appropriate function space for test and trial functions has to take the form of the product space $\mathcal{V}_B := \mathcal{V}_r^1 \times \mathcal{V}_\theta$.

Torsionfree beam model

In the case of initially straight beams with isotropic cross-section, under the assumption of vanishing shear strains, and assuming that torsional components of distributed and discrete external moments acting on the beam are negligible, the SR beam formulation can be simplified to a torsionfree (TF) model. The following formulation was originally introduced in [10] for static problems, and extended to transient problem types in [11].

The above assumptions lead to exactly vanishing torsion only in static problems. Nevertheless, it is suggested in [11] that under the mentioned restrictions, torsional values are

“very small” also in dynamic applications. Thus, the unknown vector of rotations ψ can be removed from the system so that only the unknowns $\eta^b := \mathbf{r}$ remain. We then define the semi-linear form

$$\begin{aligned} \mathbf{a}^b(\eta^b; \delta\eta^b) &:= (\rho^b \mathbf{A}\ddot{\mathbf{r}}, \delta\mathbf{r})_B + \left(EA \frac{(\|\mathbf{r}'\| - 1) \mathbf{r}'}{\|\mathbf{r}'\|}, \delta\mathbf{r}' \right)_B \\ &+ \left(\frac{EI}{\|\mathbf{r}'\|^4} \mathbf{S}(\mathbf{r}') \mathbf{r}'', \mathbf{S}(\delta\mathbf{r}') \mathbf{r}'' + \mathbf{S}(\mathbf{r}') \delta\mathbf{r}'' \right)_B \\ &- \left(\frac{2EI}{\|\mathbf{r}'\|^6} \mathbf{S}(\mathbf{r}') \mathbf{r}'', \delta(\mathbf{r}'^T \mathbf{r}') \mathbf{S}(\mathbf{r}') \mathbf{r}'' \right)_B - \left(\mathbf{m}, \frac{\mathbf{S}(\mathbf{r}')}{\|\mathbf{r}'\|^2} \delta\mathbf{r}' \right)_B \\ &- \left[\frac{\tilde{\mathbf{m}} \mathbf{S}(\mathbf{r}') \delta\mathbf{r}'}{\|\mathbf{r}'\|^2} \right]_0^l, \end{aligned} \tag{6}$$

and the linear form

$$\mathbf{b}^b(\delta\eta^b) := (\mathbf{f}, \delta\mathbf{r})_B + \left[\tilde{\mathbf{f}}^T \delta\mathbf{r} \right]_0^l. \tag{7}$$

The removal of rotational unknowns from the system of PDEs through enforcement of the TF constraint comes with the cost of additionally required smoothness of the centerline curves, namely $\mathbf{r} \in \mathcal{V}_B := \mathcal{V}_r^2$.

Navier–Stokes equations

To model the fluid field, the instationary, incompressible Navier–Stokes equations for Newtonian fluids on fixed meshes are used.

In the following, $H_0^1(\Omega_f)$ represents the standard Sobolev space on the fluid domain Ω_f with zero trace on the boundary $\partial\Omega_f$, and for a domain X we will frequently use the notation $(\cdot, \cdot)_X := (\cdot, \cdot)_{L^2(X)}$ to denote the L^2 inner product on the domain X . The fluid boundary $\partial\Omega_f = \Gamma_N^f \cup \Gamma_D^f$ can be partitioned in a Neumann boundary Γ_N^f , on which a traction \mathbf{h}^f is prescribed, and a Dirichlet boundary Γ_D^f , for which a function \mathbf{v}_D^f with prescribed velocity values on Γ_D^f is introduced. Further, let $\mathcal{V}_p = \{p \in L^2(\Omega_f) \mid \|p\|_{L^2(\Omega_f)} = 0\}$ be the space of normalized pressure solutions.

For the fluid field, we introduce the semi-linear form

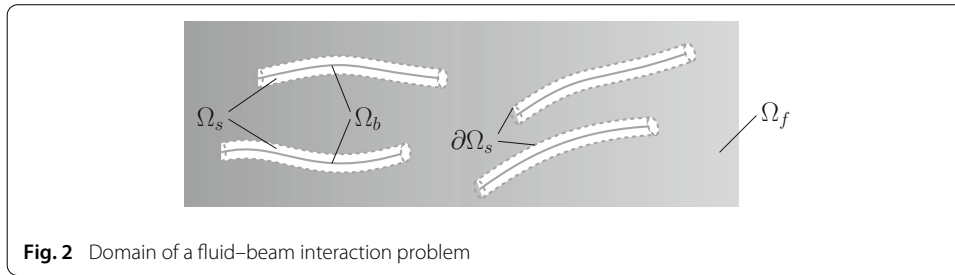
$$\begin{aligned} \mathbf{a}^f(\mathbf{v}^f, p^f; \delta\mathbf{v}^f, \delta p^f) &:= \rho^f \left(\frac{\partial \mathbf{v}^f}{\partial t}, \delta\mathbf{v}^f \right)_{\Omega_f} + 2\gamma^f (\mathcal{E}(\mathbf{v}^f), \nabla \mathbf{v}^f)_{\Omega_f} - (p^f, \nabla \cdot \delta\mathbf{v}^f)_{\Omega_f} \\ &+ \rho^f \left((\mathbf{v}^f \cdot \nabla) \mathbf{v}^f, \delta\mathbf{v}^f \right)_{\Omega_f} + (\nabla \cdot \mathbf{v}^f, \delta p^f)_{\Omega_f}, \end{aligned} \tag{8}$$

and the linear form

$$\mathbf{b}^f(\delta\mathbf{v}^f) := \rho^f (\mathbf{f}^f, \delta\mathbf{v}^f)_{\Omega_f} + (\mathbf{h}^f, \delta\mathbf{v}^f)_{\Gamma_N^f}, \tag{9}$$

with the strain rate tensor $\mathcal{E}(\mathbf{v}^f) = \frac{1}{2} (\nabla \mathbf{v}^f + (\nabla \mathbf{v}^f)^T)$, a body force \mathbf{f}^f , the dynamic viscosity γ^f , and the fluid density ρ^f , respectively.

In the case of a divergence-free initial velocity field \mathbf{v}_0^f , the behavior of the fluid on the domain Ω_f is fully described by



Problem 3 Find $(\mathbf{v}^f, p^f) \in L^2(I, H_0^1(\Omega_f)^3 + \mathbf{v}_D) \times L^2(I, \mathcal{V}_p)$, with $\mathbf{v}^f = \mathbf{v}_0$ a. e. for $t = 0$, such that

$$\int_0^T \mathbf{a}^f(\mathbf{v}^f, p^f; \delta \mathbf{v}^f, \delta p^f) - \mathbf{b}^f(\delta \mathbf{v}^f) dt = 0 \quad \forall (\delta \mathbf{v}^f, \delta p^f) \in H_0^1(\Omega_f)^3 \times \mathcal{V}_p. \quad (10)$$

Mixed-dimensional fluid–beam interaction

The single field equations introduced within the last subsections can now be used to formulate the coupled fluid–beam interaction problem. In the following, we will use the notation introduced in the last subsections and visualized in Fig. 2, and denote the fluid domain by Ω_f , the time-dependent beam domain by Ω_s , and the time-dependent beam centerline position by Ω_b . Additionally, $\partial\Omega_s$ shall represent the surface of Ω_s . From a continuum mechanics point of view, the interaction of the fluid and structure fields takes place on the shared coupling interface $\Gamma^* := \overline{\Omega_s} \cap \overline{\Omega_f}$ defined as the intersection of the respective closures of Ω_s and Ω_f . To model this interaction, the multi-physics nature of FSI problems leads to additional challenges compared to single field equations. Among these challenges is the fact, that the single field equations are based on different primal variables, i.e. the velocity field \mathbf{v}^f for the Navier–Stokes equations, and the displacement field or, more accurately in our case, the beam position \mathbf{r} for the structure.

It is thus customary to introduce the structure velocity \mathbf{v}^s in order to formulate the kinematic coupling condition

$$\mathbf{v}^f = \mathbf{v}^s \quad \text{a. e. on } \Gamma^*, \quad (11)$$

to ensure continuity of the velocity on the coupling interface Γ^* .

In the case of immersed or fictitious domain methods the fluid–structure coupling interface Γ^* is not a priori known, but has to be captured, as the embedded structures move through the fluid domain. For IBM-type methods the coupling interface is commonly defined by the surface of the structure domain moving through the fluid domain, i.e. $\Gamma^* := \partial\Omega_s$, while the fluid domain Ω_f spans the whole computational domain. Thus, for IBM-type methods, (11) leads to a surface-to-volume coupling approach, as adopted in [22, 23]. This holds in the sense that the constraints are enforced on the structure surface, while no a priori fluid surface can be defined and the coupling, thus, takes place within the fluid volume domain.

Based on the results in [20] and under the assumption of sufficiently small beam radii, the following simplifications can be argued to be valid:

- i. The coupling of the two fields can be realized on the beam centerline.
- ii. Rotational effects on the fluid flow are negligible.

For the beam velocity $\mathbf{v}^b := \dot{\mathbf{r}} : [0, l] \rightarrow \Omega_b$, (11) can be rewritten as

$$\mathbf{v}^f = \mathbf{v}^b \circ \mathbf{r}^{-1} \quad \text{a. e. on } \Omega_b, \tag{12}$$

where $\mathbf{r}^{-1} := \mathbf{r}^{-1}(t, \cdot) : \Omega_b \rightarrow [0, l]$ represents the inversion of the beam centerline curve, and serves as projection of the beam’s current geometry Ω_b in three-dimensional space onto the beam parameter space $[0, l]$. Equivalently, this condition can be written as

$$\mathbf{v}^f \circ \mathbf{r} = \mathbf{v}^b \quad \text{a. e. on } [0, l], \tag{13}$$

on the parameter space $[0, l]$.

Such a beam centerline-to-volume coupling approach, in contrast to the above mentioned surface-to-volume coupling, reduces the complexity in the computations of the coupling conditions due to the reduced dimensionality of the coupling interface, i.e. the beam centerline instead of the beam surface, in combination with the fact that no reconstruction of the beam surface is necessary.

Strong enforcement of the kinematic condition may lead to spurious effects on the fluid pressure field for immersed boundary methods because of its impact on the divergence condition. We will thus impose this condition in a weak sense. A common weak constraint enforcement technique is the Lagrange multiplier method as e.g. used in [20, 29, 30]. Application of Lagrange multipliers generally leads to a saddle point problem, which has to adhere to the inf-sup condition [31] for a stable solution. In general, for embedded finite element methods the form of such a stable Lagrange multiplier space depends on the position of the embedded mesh relative to the background mesh [32, 33]. For the case of mixed-dimensional problems, the form of such stable Lagrange multipliers is not yet well-studied. Alternatively, analogous to Nitsche’s method for classical equal-dimensional embedded finite element problems [33, 34], stabilized Lagrange multiplier methods can also be applied to mixed-dimensional embedded finite element problems [35, 36].

Nevertheless, in order to analyze the impact of this novel approach of coupling the 3D Navier–Stokes equations to 1D beam theory directly on the centerline without the need to address the additional challenge of constructing a stable Lagrange multiplier space or introducing additional stabilization terms, we refrain from using Lagrange multipliers here. Instead, we want to impose the kinematic coupling condition (12) weakly via the penalty method with a penalty parameter ϵ . Constraint enforcement via the penalty method generally leads to a more or less moderate violation of the kinematic condition. On the other hand, the method comes with the advantage of simplicity of the implementation and, more importantly, robustness with regard to stability, independently of the position of the beam. Further development to include stabilized Lagrange multipliers into the proposed method is left for future work, should challenges in connection with the penalty method, such as well-known deterioration of the condition number of the system matrix, arise.

Under the above mentioned assumption of sufficiently small beam radii, the dynamic coupling condition ensuring continuity of tractions takes the form

$$(\mathbf{f}, \delta \mathbf{r})_{\mathcal{B}} + \lim_{\partial \Omega_s \rightarrow \Omega_b} \left(\boldsymbol{\sigma}(\mathbf{v}^f) \cdot \mathbf{n}, \delta \mathbf{v}^f \right)_{\partial \Omega_s} = 0. \tag{14}$$

It is not obvious how the result of the second term looks in the limit case. However, we postulate, using the restriction operator $\boldsymbol{\Pi} : H^1(\Omega_f)^3 \rightarrow L^2(\Omega_b)$, the second term in (14) in the limit case leads to a one-dimensional integral

$$\int_{\Omega_b} \mathbf{f}_{FBI}^f(\mathbf{v}^f) \cdot \boldsymbol{\Pi} \delta \mathbf{v}^f \, ds, \tag{15}$$

where the result of the operator $\mathbf{f}_{FBI}^f(\mathbf{v}^f) : H^1(\Omega_f)^3 \rightarrow L^2(\Omega_b)$ represents a line force acting on the fluid. For the treatment via the penalty method, postulating the existence of such a line force suffices. Nevertheless, for coupling via a range of regularized Lagrange multiplier methods such as Nitsche type methods, further analysis of the exact form of the resulting line force is necessary. This question is closely related to the existence of a sufficiently smooth restriction operator $\mathbf{\Pi}$.

Remark 4 Here, $\mathbf{\Pi}$ is necessary, since the integral over the one-dimensional curve Ω_b is obviously not well-defined for $\delta\mathbf{v}^f \in H_0^1(\Omega_f)^3$. In contrast to well-known trace theorems such as [37], which postulate existence of such a trace operator on smooth boundaries of codimension one, existence conditions on the restriction operator $\mathbf{\Pi}$ in the context of a greater dimensionality gap are not yet well-studied [38]. As one of the first publications addressing the lack of trace-type theorems for mixed-dimensional problems with codimension two, Kuchta et al. [35] show sufficient regularity of such a restriction operator in the context of a mixed-dimensional model problem via averaging over a three-dimensional domain around the embedded manifold.

Even though a theoretical analysis of the well-posedness of the continuous problem in weak form would certainly represent a firm basis for further work, we will refrain here from any theoretical existence or regularity analysis. Instead, we will validate the choice of the coupling domain and illustrate its challenges by selected numerical examples in “Numerical examples”.

Finally, we introduce constraint (13) with the penalty method, as shown in [39], using a penalty contribution that increases linearly with respect to the constraint violation, by building the derivatives of an abstract quadratic penalty functional:

$$\frac{\epsilon}{2} \int_0^l (\mathbf{\Pi}\mathbf{v}^f \circ \mathbf{r} - \mathbf{v}^b) \cdot (\mathbf{\Pi}\mathbf{v}^f \circ \mathbf{r} - \mathbf{v}^b) \, ds. \tag{16}$$

Here, the slope of the linear penalization term depends on the value of the penalty parameter ϵ . This leads to the full definition of the coupled problem

Problem 5 Find $(\mathbf{v}^f, p^f, \eta^b) \in L^2(I, H^1(\Omega_f)^3 + \mathbf{v}_D) \times L^2(I, \mathcal{V}_p) \times L^2(I, \mathcal{V}_B)$, with $\mathbf{v}^f = \mathbf{v}_0$, $\eta^b = \eta_0$ a. e. for $t = 0$, and $\|\mathbf{r}'(0, \cdot)\|_{\mathbb{R}^3} = 1$ a. e. on $[0, l]$, such that

$$\begin{aligned} \int_0^T \mathbf{a}^f(\mathbf{v}^f, p^f; \delta\mathbf{v}^f, \delta p^f) - \mathbf{b}^f(\delta\mathbf{v}^f) + \epsilon \int_0^l (\mathbf{\Pi}\mathbf{v}^f \circ \mathbf{r} - \mathbf{v}^b) \cdot \mathbf{\Pi}\delta\mathbf{v}^f \circ \mathbf{r} \, ds \, dt = 0, \\ \int_0^T \mathbf{a}^b(\eta^b; \delta\eta^b) - \mathbf{b}^b(\delta\eta^b) - \epsilon \int_0^l (\mathbf{\Pi}\mathbf{v}^f \circ \mathbf{r} - \mathbf{v}^b) \cdot \delta\mathbf{r} \, ds \, dt = 0, \end{aligned} \tag{17}$$

for all $(\delta\mathbf{v}^f, \delta p^f, \delta\eta^b) \in H_0^1(\Omega_f)^3 \times \mathcal{V}_p \times \mathcal{V}_B$.

Discretization and numerical integration

The PDEs of both fields introduced in “Problem formulation” are spatially discretized using the FEM, and the evolution in time is modeled using a finite difference time stepping scheme. We want to briefly recount particular challenges arising for the respective fields,

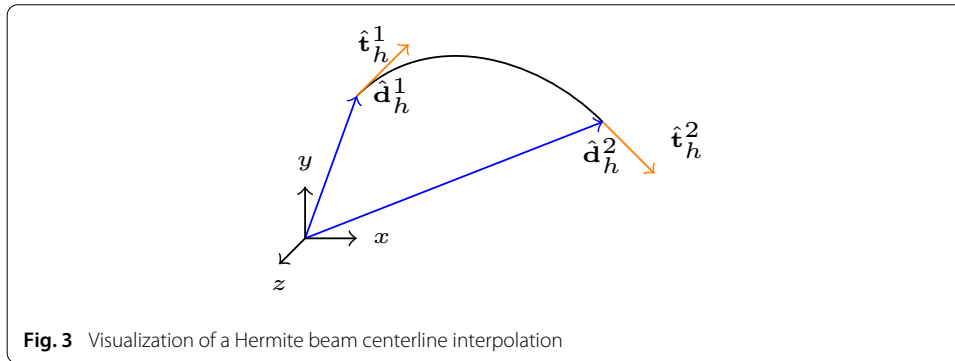


Fig. 3 Visualization of a Hermite beam centerline interpolation

and point out how they were overcome within the present work. Afterwards, the numerical treatment of the coupling terms will be presented in more detail.

Fluid field

Numerical treatment of the incompressible Navier–Stokes equations presented in “Navier–Stokes equations” with the FEM leads to a mixed element formulation, which generally has to adhere to the inf-sup stability condition, cf. [31]. Nevertheless, we will take the alternative approach and circumvent the inf-sup condition by stabilizing linear equal-order hexahedral finite elements using the PSPG/SUPG stabilization method with an additional div-grad stabilization term [40,41]. Here, only the PSPG method actually circumvents the inf-sup condition. The SUPG method prevents numerical instabilities in convection-dominated flows, while the grad-div term improves well-posedness as well as solution accuracy of the Navier–Stokes equations. A detailed description and recounting of all stabilization terms within the used implementation can be found in [42].

Thus, the unknown fields of Problem 3 are discretized as

$$\mathbf{v}_h^f = \sum_{k=1}^{n^f} N_k \hat{\mathbf{v}}_h^{f,k} \quad \text{and} \quad p_h^f = \sum_{k=1}^{n^f} N_k \hat{p}_h^{f,k}, \tag{18}$$

where n^f is the number of fluid nodes, the N_k represent the linear shape functions, and $\hat{\mathbf{v}}_h^{f,k}$ and $\hat{p}_h^{f,k}$ denote the nodal degrees of freedom of the velocity and pressure at node k , respectively.

To discretize the Navier–Stokes equations in time, a one-step- θ time stepping scheme is employed for the examples in “Numerical examples”.

Beams

The additionally required smoothness of the centerline representation within the torsion-free beam formulation leads to the necessity of C^1 -continuous shape functions. This is realized by using 3rd-order Hermite shape functions for the beam centerline, as described in [9]. Even though these C^1 -continuous shape functions are only necessary for the torsionfree beam element, we will also apply them to the centerline interpolation of the SR finite element used in the numerical examples in “Numerical examples”. As shown in Fig. 3, this leads to a two-noded element with six centerline degrees of freedom (DoFs) per node and the shape functions H_k^d for the nodal positions $\hat{\mathbf{d}}_h$ in all three dimensions, and H_k^t for the nodal tangents $\hat{\mathbf{t}}_h$ in all three dimensions.

The description of the discrete beam position thus takes the form

$$\mathbf{r}_h = \sum_{k=1}^{n^b} H_k^d \hat{\mathbf{d}}_h + \frac{l_{ele}}{2} \sum_{k=1}^{n^b} H_k^t \hat{\mathbf{t}}_h, \tag{19}$$

with the number of beam-centerline nodes n^b and the initial beam element length l_{ele} .

As argued in “[Mixed-dimensional fluid–beam interaction](#)”, for relatively slender beams, it is valid to neglect the effect of beam rotations on the fluid flow. Thus, only the centerline terms, which are independent of the employed beam theory, appear in the coupling equations. Therefore, we skip the details of the cross-section rotation discretization here. The interested reader may be referred to [11] for details on the applied Petrov-Galerkin method for discretization in space, and to [43,44] for the Lie-group Generalized- α time stepping scheme based on multiplicative updates, which is applied to the rotational degrees of freedom of the employed beam finite elements.

The time evolution of the beam centerline position for all examples in “[Numerical examples](#)”, including SR as well as torsionfree beam elements, is discretized using a Generalized- α time integration scheme [45] with the spectral radius set to $\rho_\infty = 1$, and all parameters set accordingly to produce a second-order accurate time stepping scheme.

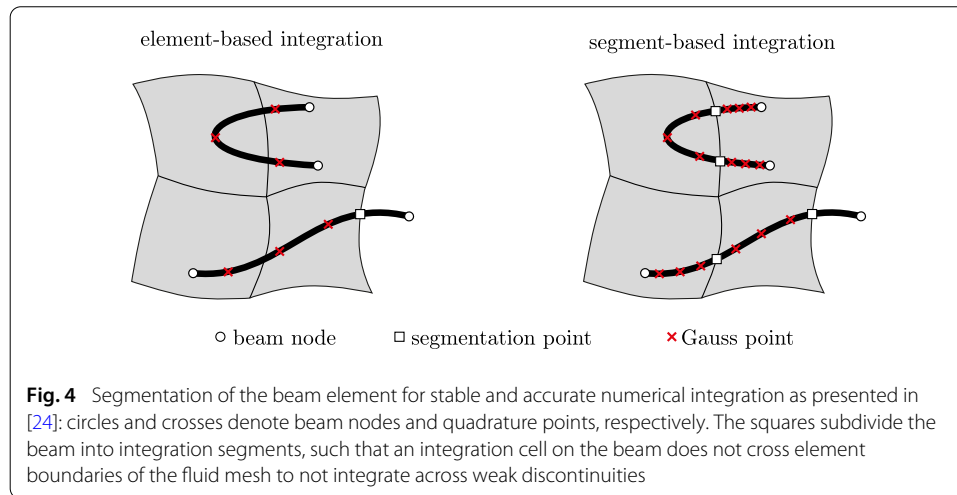
Numerical treatment of the coupling condition

After discretization of both, the three-dimensional incompressible isothermal Navier–Stokes equations and the beam formulation, two distinct non-matching meshes emerge. The fixed fluid mesh takes on the role of a background mesh. The beam mesh, oriented along the beams’ centerlines, is superimposed, leading to the use of an embedded mesh approach. The proposed coupling discretization can be characterized as a Gauss-point-to-segment (GPTS) approach, as e.g. commonly used in contact mechanics. Here, the constraint equation (13) is evaluated at each quadrature point of the beam. Future work will focus on the extension to a segment-to-segment (STS) based approach. In our previous work [24], it was shown that realizations of the mesh tying of a beam to a solid volume can benefit from such a STS based approach in terms of well-posedness and the avoidance of locking effects. Nevertheless, the numerical examples in “[Numerical examples](#)” section will show that the used GPTS approach is more than sufficient for a multitude of application problems.

In the following, we will summarize the beam element shape functions at node k , including positional shape functions H_k^d and tangential shape functions H_k^t , as H_k . Inserting (19) and (18) into the coupling contributions introduced in Problem 5 gives rise to four blocks of coupling contributions to the discretized system:

$$\begin{aligned} \epsilon_{\mathbf{KB}\mathbf{B}}(k, j) &:= \epsilon \int_0^l H_k H_j \, ds, & \epsilon_{\mathbf{KF}\mathbf{F}}(k, j) &:= \epsilon \int_0^l (N_k \circ \chi) (N_j \circ \chi) \, ds, \\ \epsilon_{\mathbf{KB}\mathbf{F}}(k, j) &:= \epsilon_{\mathbf{KF}\mathbf{B}}(j, k) := \epsilon \int_0^l H_k (N_j \circ \chi) \, ds, \end{aligned} \tag{20}$$

with χ denoting the projection of a point in the parameter space of the beam centerline to the corresponding point in the parameter space of the discretized fluid volume. Subscripts \mathbf{BB} and \mathbf{FF} denote beam and fluid contributions, respectively. Mixed subscripts \mathbf{FB} and \mathbf{BF} refer to the mixed coupling interaction contributions. Note that here, the projection

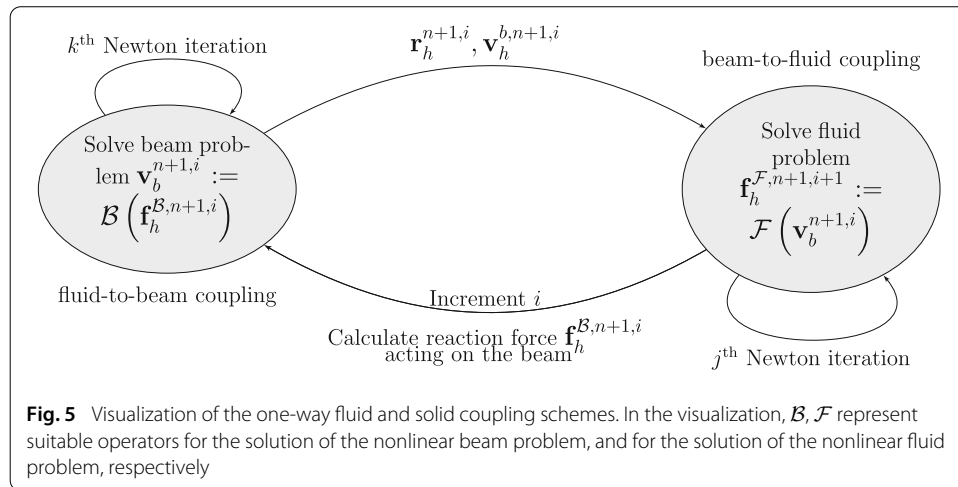


χ is achieved by computing the current position \mathbf{r}_h at each Gauss point of the discretized beam centerline curve. Thus, the projection χ introduces a hidden dependency on the current deformation of the beam, and adds an additional nonlinearity to the system. Within a monolithic solution scheme, the addition of the above coupling matrices to the system matrix would be straightforward based on the continuous formulation in Problem 5. However, we have chosen to use a partitioned FSI solution scheme. The exact way the coupling matrices in (20) are added to the numerical system, as well as details on the algorithmic treatment of the resulting nonlinearity, will be given in “One-way coupling algorithms”.

In order to deal with weak discontinuities at the fluid element boundaries as they arise during element-wise numerical integration of the integrals in (20), a more accurate segment-based integration approach to evaluate (20) is adopted from our previous work [24] and visualized in Fig. 4.

As the present problem is transient, the time discretization for the single fields is also applied to the constraint. Aiming at a temporally consistent exchange of coupling information [30,46], this leads to a linear combination of the coupling contributions $\mathbf{K}_{\circ\circ}^n$, evaluated at the old time step n , and $\mathbf{K}_{\circ\circ}^{n+1}$, at the new time step $n + 1$, depending on the time integration parameters for the employed fluid and structure time integration schemes, respectively. Further information on the algorithmic details will be given in the upcoming section.

Remark 6 Note that the assumption of relatively small beam radii also extends to the fluid element sizes. The choice of enforcing the fluid–beam coupling on the beam centerline introduces a singularity to (17). The fluid elements need to be large enough to not be able to resolve this singularity, effectively “smearing” the interface over the volume of the element as is common in classical immersed methods. The exact domain of applicability of the method with respect to the quotient of beam diameter and fluid element size also depends on the properties of the simulated fluid. The influence of the background mesh resolution on the fluid velocity resolution will be studied in section Numerical examples.



One-way coupling algorithms

The solution algorithm is set up as a partitioned FSI algorithm with the counter i representing the coupling iteration. The long-term goal of our current work is the application of the proposed mixed-dimensional coupling approach to a fully coupled fluid–beam interaction problem. Nevertheless, in order to analyze the implications and limitations stemming from the numerical embedding of a reduced-dimensional structure in three-dimensional flow, here, we first want to introduce two one-way coupled algorithmic variants. On the one hand, this analysis serves as a validation of the different components, which can then be combined to a fully two-way coupled fluid–beam interaction algorithm. On the other hand, as will be shown in “Numerical examples”, there exist numerous application scenarios, in which the one-way coupling variants proposed here represent valid models at rather low computational complexity. This highlights their value not only as a work-in-progress, but particularly as stand-alone models for a broad spectrum of practically relevant examples.

In the upcoming subsections, we will explain the implications for the two different one-way coupling schemes depicted in Fig. 5. Nevertheless, some details are valid for both variants. Firstly, the single fields are solved using a Newton–Raphson algorithm. The number of Newton iterations within the single fields will be represented by the counter j and k for the fluid and structure field, respectively. Secondly, to capture the exact FBI boundary, i.e. the position of the beam centerline, an octree-based search is performed after each change in the state of the solution of the nonlinear beam problem. This way, all fluid element nodes in a prescribed vicinity of the displaced beam element nodes are found. The segmentation procedure visualized in Fig. 4 is applied to all resulting pairs of intersecting beam and fluid elements. If a valid segment is found within such a pair, its contributions are added to the respective matrices in (20). Otherwise, the pair does not contribute to the coupling terms.

In order to not complicate the notation unnecessarily, we will drop the iteration counter indices i , for the partitioned algorithm, and j as well as k , for the field-specific Newton–Raphson solver, to indicate independence of a variable with respect to that index.

One-way coupling for rigid beams

First, we analyze the special case of immersed rigid beams. Assuming that the effect of the fluid flow on the beam is negligible allows to use a one-way coupling algorithm which only affects the fluid partition. In this coupling variant, the partitioned algorithm will converge after only one iteration leading to a simple staggered algorithm in which each field is only solved once. In other words, the FSI iteration index i in Fig. 5 can be dropped. We will denote this one-way coupling scheme as beam-to-fluid coupling variant within the remaining part of the paper.

Motivated by classical Dirichlet–Neumann partitioned schemes, where Ω_f acts as Dirichlet and Ω_b as Neumann Partition, the discrete penalty constraint at time step $n + 1$

$$\epsilon \mathbf{K}_{\mathcal{FF}}^{n+1} \mathbf{v}_h^{f,n+1} = \epsilon \mathbf{K}_{\mathcal{FB}}^{n+1} \mathbf{v}_h^{b,n+1}, \tag{21}$$

is enforced on the fluid partition for a prescribed discrete beam velocity $\mathbf{v}_h^{b,n+1}$. Here, the coupling variable $\mathbf{v}_h^{b,n+1}$ is computed by the beam’s time integration scheme. Constraint (21) can be interpreted as the weak enforcement of the Dirichlet constraint

$$\mathbf{v}_h^{f,n+1} = \tilde{\mathbf{v}}_h^{b,n+1} := \left(\mathbf{K}_{\mathcal{FF}}^{n+1} \right)^{-1} \mathbf{K}_{\mathcal{FB}}^{n+1} \mathbf{v}_h^{b,n+1}, \tag{22}$$

where $\tilde{\mathbf{v}}_h^{b,n+1}$ can be interpreted as the projection of the beam velocity onto the fluid mesh. Note that the matrices $\mathbf{K}_{\mathcal{FF}}^{n+1}$ and $\mathbf{K}_{\mathcal{FB}}^{n+1}$ in (21) only depend on the geometry of the overall system. Because the fluid is solved in a purely Eulerian frame, the above mentioned coupling matrices are constant within the nonlinear solution step of the fluid partition. Accounting for the time discretization, the following equation is added to the fluid system in iteration step $j + 1$ of the Newton–Raphson method:

$$\begin{aligned} \theta \epsilon \mathbf{K}_{\mathcal{FF}}^{n+1} \Delta \mathbf{v}_h^{f,n+1,j+1} &= \theta \left(\epsilon \mathbf{K}_{\mathcal{FB}}^{n+1} \mathbf{v}_h^{b,n+1} - \epsilon \mathbf{K}_{\mathcal{FF}}^{n+1} \mathbf{v}_h^{f,n+1,j} \right) \\ &+ (1 - \theta) \left(\epsilon \mathbf{K}_{\mathcal{FB}}^n \mathbf{v}_h^{b,n} - \epsilon \mathbf{K}_{\mathcal{FF}}^n \mathbf{v}_h^{f,n} \right). \end{aligned} \tag{23}$$

One-way coupling for light fibers

Conversely, we analyze the special case of freely moving, light fibers, for which the effect on the fluid flow can be neglected. Within the remainder of this paper, we will refer to this one-way coupling scheme as the fluid-to-beam coupling variant. Within this coupling variant, generally multiple FSI iterations per time step are necessary to converge towards an equilibrium between the two separate fields. The reasons for multiple FSI iterations within a one-way coupling algorithm are explored after introducing the manner in which the coupling is applied to the beam system.

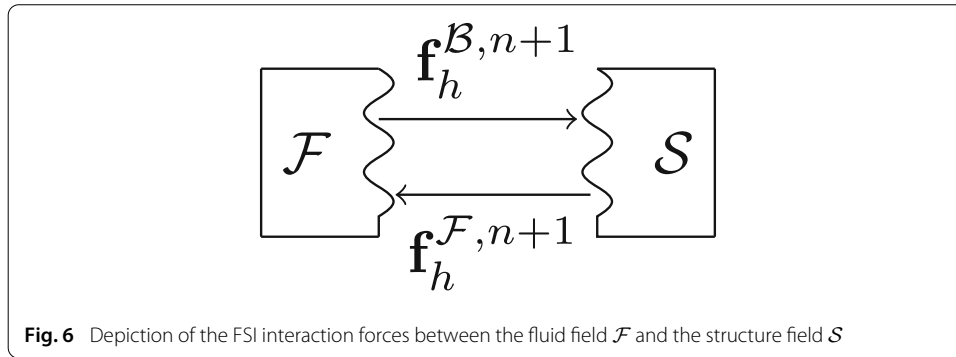
Once more, motivated by classical Dirichlet–Neumann partitioned schemes, where the fluid domain is treated as Dirichlet partition, while the structure domain takes on the role of the Neumann partition, the coupling condition (13) is now applied to the beam in the form of the discrete interaction force

$$\mathbf{f}_h^{\mathcal{BS},n+1,i+1} := \epsilon \mathbf{K}_{\mathcal{BB}}^{n+1,i+1} \mathbf{v}_h^{b,n+1,i+1} - \epsilon \mathbf{K}_{\mathcal{BF}}^{n+1,i+1} \mathbf{v}_h^{f,n+1}, \tag{24}$$

within every iteration $i + 1$ of the partitioned algorithm. For the employed generalized- α time integration scheme, this, in turn, leads to the contribution

$$\left(1 - \alpha^f \right) \mathbf{f}_h^{\mathcal{BS},n+1,i+1} + \alpha^f \mathbf{f}_h^{\mathcal{BS},n}, \tag{25}$$

to the residual of the structure system. Due to the partitioning, the applied interaction force is constant during the solution of the structure problem, which leads to a neglect



of the change in geometry of the beam, and therefore, the FSI interface. This nonlinearity is thus not treated within the Newton solver for the beam system, but successively updated through i FSI iterations between the two fields. The handling of the kinematic coupling constraint as a Neumann condition within a fixed-point iteration enables a flexible choice of the beam’s time integration scheme as compared to a full Newton method, in which the velocity-displacement relationship would enter the linearization of the problem. Nevertheless, for very slender structures, among others, generally convergence issues of the partitioned scheme associated with the well-known added mass effect may arise. More detailed characterizations of the added-mass effect, and a comprehensive study on methods to overcome the arising challenges, can be found in [47,48]. In our case, the convergence behavior of the fluid-to-beam coupling variant is improved via the Aitken relaxation introduced in [49]. For this one-way coupling scheme, we will relax the force applied to the beam instead of the solid displacement increment and consequently the velocity given to the fluid partition. This is in contrast to the method outlined in [50] for typical applications of classical surface-coupled FSI algorithms and can be motivated by the following fact: in typical FBI examples, due to the beam’s slender geometry and, thus, its susceptibility to small forces, the beam solution is more sensitive to changes in the geometric configuration than the fluid solution.

Remark 7 We want to give a few remarks on the connection between classical Dirichlet–Neumann algorithms and the algorithm proposed here. Transformation of (17) leads to

$$\epsilon \int_0^L (\boldsymbol{\Pi} \mathbf{v}^f \circ \mathbf{r} - \mathbf{v}^b) \cdot \boldsymbol{\Pi} \delta \mathbf{v}^f \circ \mathbf{r} \, ds = \mathbf{b}^f(\delta \mathbf{v}^f) - \mathbf{a}^f(\mathbf{v}^f, p^f; \delta \mathbf{v}^f, \delta p^f), \quad (26)$$

meaning that, for the forces to be in equilibrium within each time step,

$$\mathbf{f}_h^{\mathcal{F},n+1} := \epsilon \mathbf{K}_{\mathcal{F}\mathcal{F}}^{n+1} \mathbf{v}_h^{f,n+1} - \epsilon \mathbf{K}_{\mathcal{F}\mathcal{B}}^{n+1} \mathbf{v}_h^{b,n+1} \quad (27)$$

will take on the additional force applied to the fluid because of the FBI constraint.

As depicted in Fig. 6, in the present case of non-matching meshes, $\mathbf{f}_h^{\mathcal{B},n+1}$ represents the reaction force to $\mathbf{f}_h^{\mathcal{F},n+1}$ acting on the beam mesh. Thus, the load and motion transfer scheme introduced above for one-way coupling schemes leads to a weak Dirichlet–Neumann partitioned algorithm in the case of two-way coupling. Here, the Dirichlet conditions are weakly imposed on the fluid partition while the beam domain takes on the role of the Neumann partition.

Nevertheless, this interpretation of $\mathbf{f}_h^{\mathcal{B},n+1}$ as the FBI interaction force acting on the beam weakens in the case of one-way coupling when $\mathbf{f}_h^{\mathcal{F},n+1}$ is assumed to be negligible. In this case, (24) should be interpreted as weak enforcement of the Dirichlet constraint (13), for which the nonlinearity introduced by the change in geometry is treated by the Aitken relaxation instead of a Newton method. Both interpretations will be further addressed in “Numerical examples” section.

Remark 8 Even if the assumptions of a perfectly rigid beam might not hold for a given physical setup, the numerical model of the beam-to-fluid coupling variant introduced in “One-way coupling for rigid beams” still represents a physically meaningful and solvable situation. This is not necessarily the case for the fluid-to-beam coupling variant introduced in “One-way coupling for light fibers”: as argued in Remark 7, the penalty force does not represent the FSI force in the fluid-to-beam coupling case. Instead, it is simply the scaled negative constraint violation. Since the beam is invisible to the fluid field, violation of the assumption of a perfectly soft, freely movable, light fiber may lead not only to unphysically large forces acting on the beam but also to a deterioration of convergence behavior of the given algorithm for large penalty parameters due to the ill-posedness of the underlying continuous problem. It is thus paramount to carefully check the modeling assumptions of this one-way coupling variant before considering applying the algorithm. The applicability of the proposed approach under no or even moderate violation of the fluid-to-beam coupling assumptions will be demonstrated in “Numerical examples”.

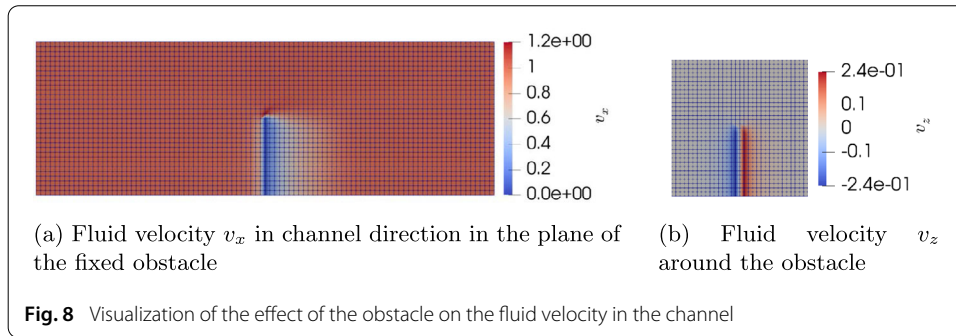
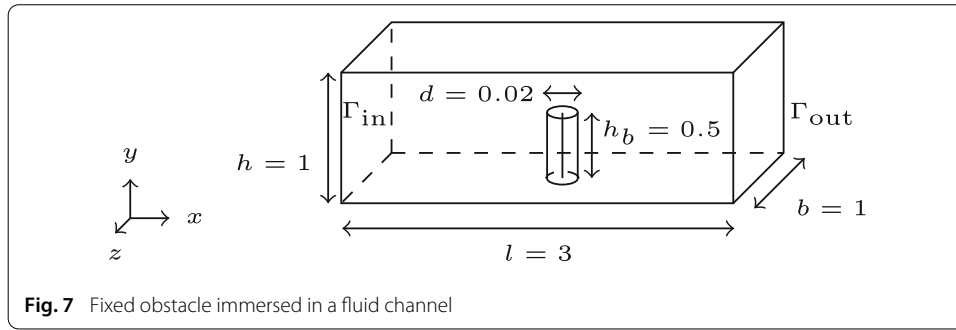
Numerical examples

The following numerical examples are chosen to illustrate the capabilities of the proposed fluid–beam interaction approach. To this end, this section focuses on the special cases of very stiff slender bodies and very light fibers in order to analyze basic properties of the one-way coupling cases. If not stated differently, all simulations include torsionfree beam elements. For all examples, the fluid is assumed to be initially at rest. All models are set up using the pre-processor MeshPy [51], and the simulations are performed with the in-house multi-physics research code BACI [52].

Fixed obstacle immersed in a fluid channel

The purpose of the first example is twofold: its simplicity allows to easily compute the solution of a full 3D simulation to validate the proposed mixed-dimensional coupling approach, and to study the convergence behavior of the resulting numerical error with respect to uniform mesh refinement. Secondly, the example is used to illustrate the dependence of the constraint violation within the fluid partition on the chosen penalty parameter. While numerical constraint enforcement via the penalty method is common practice in constrained optimization problems as well as applications such as contact dynamics, a use within computational fluid dynamics as presented within this work is not as common. This novelty warrants a closer look at the effect of this choice of constraint enforcement method.

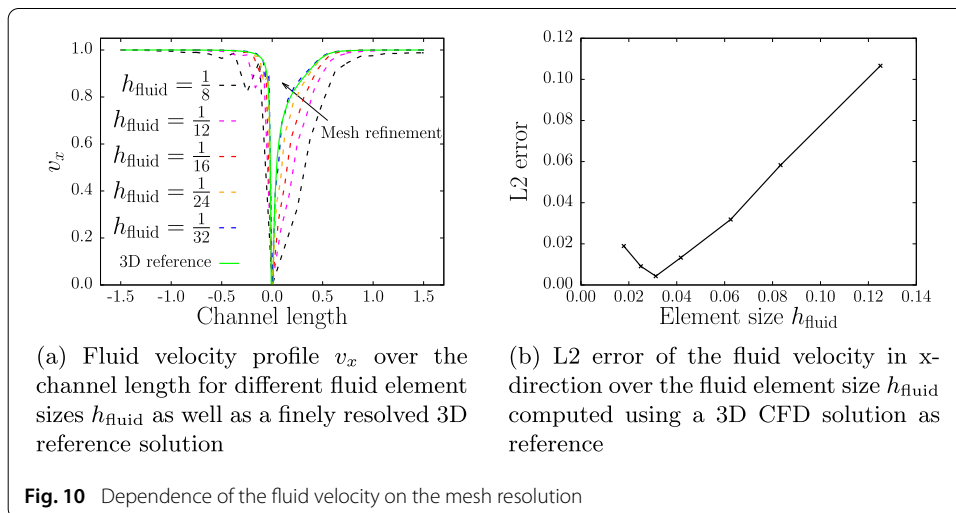
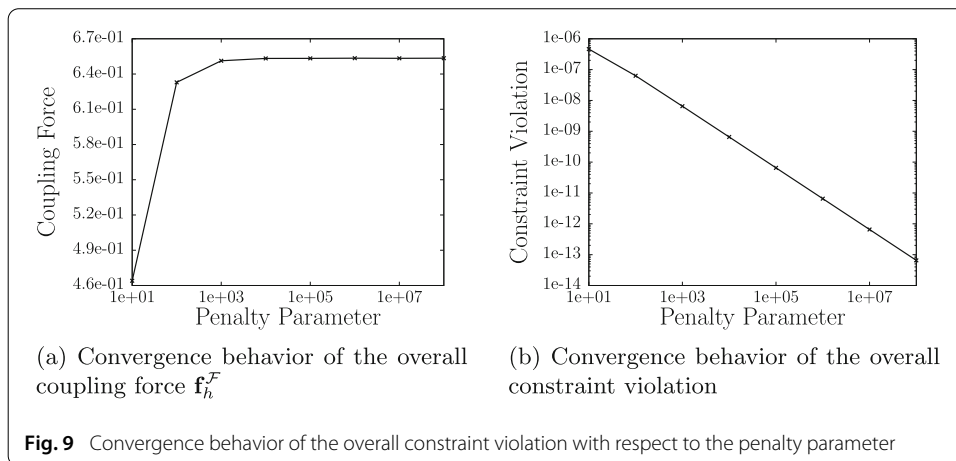
Figure 7 shows the configuration of a fixed obstacle immersed in a mono-directional fluid flow. The time-dependent inflow velocity in x -direction $v_{in} = 0.5 \cdot (1 - \cos(10\pi t))$ for $t \in [0, 0.1]$, and $v_{in} = 1$ for $t > 0.1$, is prescribed on the surface Γ_{in} . On the surface Γ_{out} , a zero-traction condition is applied and perfect-sliding conditions are enforced on



all other surfaces. The fixed beam obstacle is placed in the middle of the lower channel surface and is assumed to not be affected by the surrounding fluid. This allows to consider only one-way coupling from the fixed beam onto the fluid. The fluid is assumed to have a density of $\rho = 1$ and a dynamic viscosity of $\mu = 0.004$. The fluid domain is discretized by $96 \times 32 \times 32$ stabilized 8-noded hexahedral finite elements with equal-order interpolation, and the evolution in time is solved by the Crank–Nicolson method, i.e. the one-step- θ method with $\theta = \frac{1}{2}$, with time step size $\Delta t = 10^{-3}$. Note that for the proposed approach therefore neither the radius of the beam obstacle nor its material properties enter the simulation. Instead, only the effect of the applied Dirichlet conditions on the fluid solution in dependence on the penalty parameter shall be analyzed.

The obstacle is expected to decelerate the fluid in its vicinity to a resting state, and thereby redirect the fluid flow to the sides and to the top. For a penalty parameter $\epsilon = 10^4$, simulation results illustrating this behavior are shown in Fig. 8. The deceleration in x -direction as well as the deflection of the flow velocity in z -direction is clearly visible. In the regarded case of a fixed beam, the constraint violation after time step $n + 1$ is measured by calculating the L_2 -norm of $\mathbf{K}_{\mathcal{F}\mathcal{F}}^{n+1} \mathbf{v}_h^{f,n+1}$. Figure 9b shows the dependence of the constraint violation on the penalty parameter at time $t = 0.5$ when a steady state solution is reached.

In addition, Fig. 9a allows a closer look at the L_2 -norm of the computed penalty force $\mathbf{f}_h^{\mathcal{F},n+1} = \epsilon \mathbf{K}_{\mathcal{F}\mathcal{F}}^{n+1} \mathbf{v}_h^{f,n+1}$ acting on the fluid at time $t = 0.5$. In “inlinkOne-way coupling algorithms sec:algo”, it was argued, that this penalty force will take on the amount of force necessary to fulfill the given FBI constraint, and thus can be interpreted as the interface force acting onto the fluid. For this hypothesis to be valid, the penalty force needs to be practically independent of the chosen value of the penalty parameter, or equivalently, linear convergence of the constraint violation $\mathbf{K}_{\mathcal{F}\mathcal{F}}^{n+1} \mathbf{v}_h^{f,n+1}$ towards 0 is expected as the penalty parameter increases. Figure 9 suggests that both assumptions hold true for sufficiently large values of the penalty parameter. This is also an important basis for further



work towards a fully coupled FBI framework, where the position of the beam centerline, and thus the geometry of the coupled problem, as well as the beam velocity, depend on the penalty force.

Next, the behavior of the 1D–3D coupling solution with respect to the fluid mesh resolution shall be analyzed. Figure 10a shows the space-dependent steady state solution of \mathbf{v}_x along the x -axis for different mesh resolutions. Here, the reference solution is computed using a full 3D computational fluid dynamics (CFD) simulation, in which the 3-dimensional beam domain is cut out of the fluid domain, and no-slip boundary conditions are enforced on the newly generated beam surface. The fluid field is discretized using 641.928 DoFs. This 3D reference solution is used to compute the L2 error of the shown 1-dimensional velocity profile plotted over the fluid element size h_{fluid} in Fig. 10b.

Examining the fluid velocity profile over the channel length plotted in Fig. 10a, the fluid flow is slowed down by the beam, while the flow is undisturbed far from the obstacle as expected. The kink in the FBI solutions just before the beam stems from the neglect of capturing the exact behavior of the pressure solution at the beam. As investigated by Baaijens in [20], the pressure would exhibit a jump at the beam, which cannot be represented within the discretization space of continuous piece-wise linear functions. Using appro-

appropriate discretization spaces allowing for pressure jumps on element boundaries as in [20] or even enhancing the space as in the extended finite element method in [42] can potentially solve this problem. We deliberately refrain from such advanced fluid discretizations, since we are targeting an efficient solver for macroscopic effects in FBI, not a fine-scale resolution in the vicinity of the coupling interface. For now, note that refinement of the mesh near the obstacle can smooth these kinds of kinks.

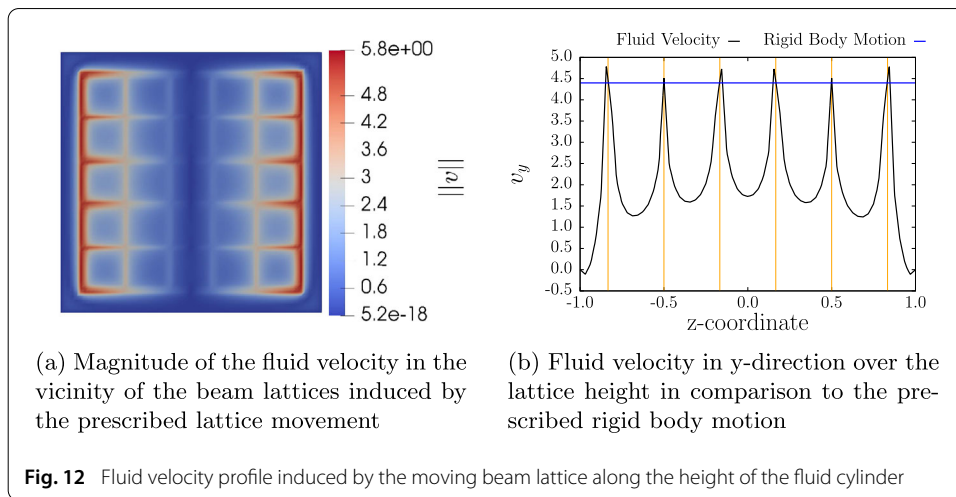
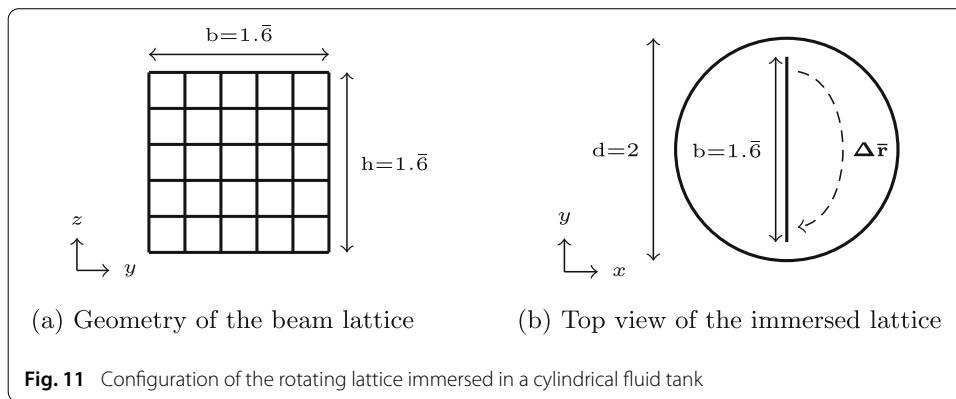
Further, it becomes evident that the fluid profile computed with the FBI method at a fluid element size of $h_{\text{fluid}} = \frac{1}{32}$ matches the 3D reference solution very well. As a more quantitative analysis, the L^2 error of the 1D profiles with respect to the 3D CFD solution is shown in Fig. 10b. The 1D–3D coupling approach exhibits a linear convergence behavior with respect to uniform mesh refinement, which is expected due to the fact that the beam radius does not enter the simulation, and instead, the area in which the fluid is slowed down is scaled by the fluid element size. This convergence behavior as well as the error values are in line with the results for general academic mixed-dimensional models reported in [35, 36, 53], none of which exceeded linear convergence in the primary variable of the 3-dimensional field. It can also be seen, that for the proposed FBI approach, further mesh refinement of the fluid background mesh will lead to a narrowing of the affected fluid area and, thus, a growing error with respect to the 3D reference solution as evident in Fig. 10b. This convergence behavior closely resembles the one that has been observed and discussed for a similar approach applied to mixed-dimensional solid-beam coupling in [24]. Such a behavior is to be expected, as the proposed 1D–3D coupling approach is only valid under the assumptions of relatively small beam radii compared to the fluid element size as elaborated in Remark 6. For problem setups, in which the model assumptions are violated, no further convergence can be expected and a different modeling technique should be chosen.

In conclusion, this example demonstrates that the FBI solution converges towards the reference solution within a spectrum of fluid element sizes, for which the model assumptions, as discussed in Remark 6, are fulfilled. While recovery of optimal convergence for 1D–3D coupling approaches is still a worthwhile topic of ongoing research, the matching of the solutions obtained with mixed-dimensional and fully resolved models satisfies the goals of this work and validates the general applicability of the proposed FBI model.

Rotating lattice

After analyzing the behavior of the constraint violation itself, we now want to observe the global effects of slender bodies on fluid flow as well as the proposed method's robustness under large displacements. To this end, we will regard a rigid lattice of dimension 1.6×1.6 made up of beams, as depicted in Fig. 11a, immersed in a cylindric tank filled with fluid at rest. The cylinder has a radius of 1, and a height of 2 in order to encompass the entire beam lattice. Furthermore, all surfaces are modeled with no-slip boundary conditions. The fluid is assumed to have a density of 1.0 and a dynamic viscosity of 4, while the beams are again assumed to not be affected by the fluid itself. The lattice starts rotating around the vertical axis of the cylinder as depicted in Fig. 11b. The movement is described by the rigid body motion

$$\Delta \bar{\mathbf{r}} = \begin{pmatrix} \cos(2 \cdot \pi \cdot \bar{\nu}) \cdot x - \sin(2 \cdot \pi \cdot \bar{\nu}) \\ \sin(2 \cdot \pi \cdot \bar{\nu}) \cdot x + \cos(2 \cdot \pi \cdot \bar{\nu}) \end{pmatrix}, \quad (28)$$

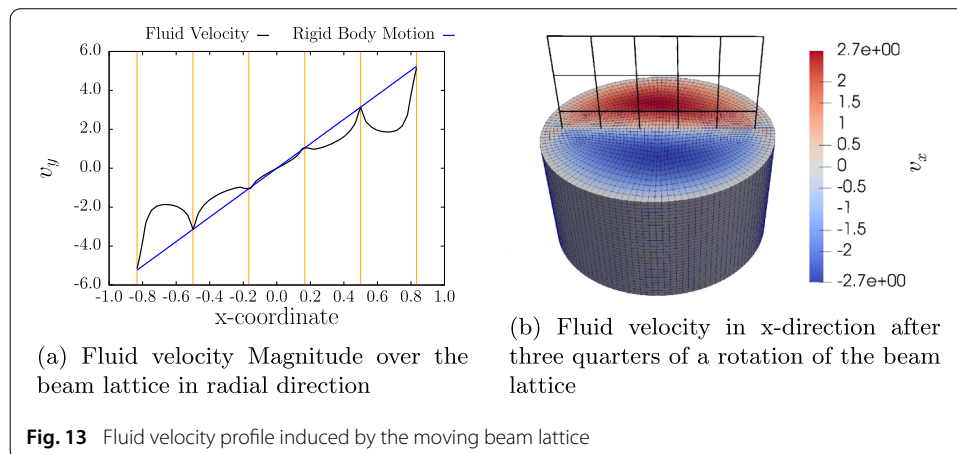


with the time-dependent scaling factor $\bar{v} = 0.5 \cdot (1 - \cos(2\pi t))$ for $t \in [0, 0.5]$, and $\bar{v} = 1$ for $t \geq 0.5$. For this example, the time step is chosen as $\Delta t = 10^{-3}$, the evolution in time is discretized with the Backward Euler time stepping scheme, and the FBI constraint is enforced with a penalty parameter of $\epsilon = 100$. The entire fluid domain is discretized with 114,688 hexahedral finite elements arranged into 64 layers along the height of the cylinder. The fluid mesh is depicted in Fig. 13b.

It is expected that the beam lattice incites the fluid within the cylinder to start rotating. Herein, the fluid velocity in the vicinity of the beam matches the rigid body motion up to a penalty constraint violation, while the velocity further away from the rotator is indirectly accelerated by the surrounding fluid.

Figures 12 and 13 depict the fluid solution after three quarters of a rotation of the lattice at time $t = 0.837$. Note that in the regarded time step, the velocity of the lattice in x-direction is negligible.

Figure 12a shows that the absolute velocity $\|\mathbf{v}\|_2$ follows the lattice structure of the beam geometry. Figure 12b offers a closer look at the velocity measured over the cylinder height at the coordinate $x = 0.7$ and $y = 0$. It can be seen that the local velocity extrema of the fluid velocity v_y (black line) do not exactly match the beam positions (vertical lines). Instead, the fluid velocity in the vicinity of the beam struts is even a little higher than the prescribed rigid body velocity (blue line). This can be explained by the fact that, within the proposed coupling approach, the fluid solution is not enriched by additional



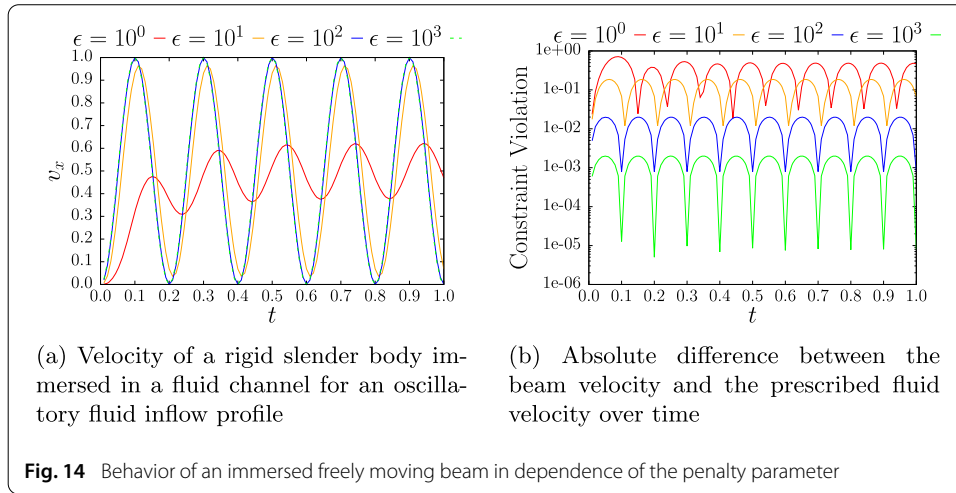
shape functions to model gradient jumps within elements as is the case for extended finite elements as used in [42]. Thus, no sudden change in the fluid velocity within an element can be represented. The local extrema of the flow velocity thus fall to the finite element node closest to the beam, as seen in 12b, and lead to a discretization error near the interface, caused by the reduced complexity of the proposed approach.

Figure 13a shows the fluid velocity profile v_y , and thus the total velocity, in radial direction along the x-axis cutting the beam lattice in a top and a bottom half. As expected from a physical point of view, it can be seen that the fluid velocity matches the rotation velocity very closely in the vicinity of the beam lattice while it lags behind within the holes of the mesh. Clearly, for this case of a less steep evolution of the fluid solution than in Fig. 12b, the discretization error due to missing enrichments of the underlying discretization space is less pronounced.

Figure 13b shows the fluid mesh, which was traversed via different cut scenarios of the overall beam lattice with the background mesh within each time step, until the presented state was reached. In the present case, the fluid mesh contains 114,688 fluid elements, while the lattice is discretized using 1200 beam elements in order to force the creation of numerous different integration segments by the segmentation procedure visualized in Fig. 4. In this case, application of the segmentation procedure leads to the creation of a number of different integration segments between 1597 and 1940 for each time step. Here, the exact number of segments for each time step depends on the position of the lattice relative to the background mesh and, thus, varies rather considerably over the course of the simulation due to the large lattice displacement and the numerous different cut scenarios stemming from the unsymmetric meshing of the fluid tank. Therefore, the regarded example also serves as a preliminary validation of the robustness of the applied segmentation procedure shown in Fig. 4 with respect to varying intersection scenarios of the embedded with the background mesh.

Light fibers in fluid flow

Having analyzed the one-way coupled case of rigid beams affecting fluid flow, this section is meant to investigate the effect of the penalty parameter on the coupling of a light fiber being transported by a fluid.



In this example, the fluid is again assumed to be contained in a hexahedron of dimensions $1 \times 1 \times 3$ as introduced in “Fixed obstacle immersed in a fluid channel”. Only now all DoFs in channel direction are set to be free and a velocity of zero is prescribed in both other directions, leading to a pseudo one-dimensional setup. In order to analyze the time-dependent behavior of the beam in dependence on the penalty parameter, an oscillatory inflow velocity

$$v_{in} = 0.5 \cdot (1 - \cos(\pi \cdot 10 \cdot t)) \tag{29}$$

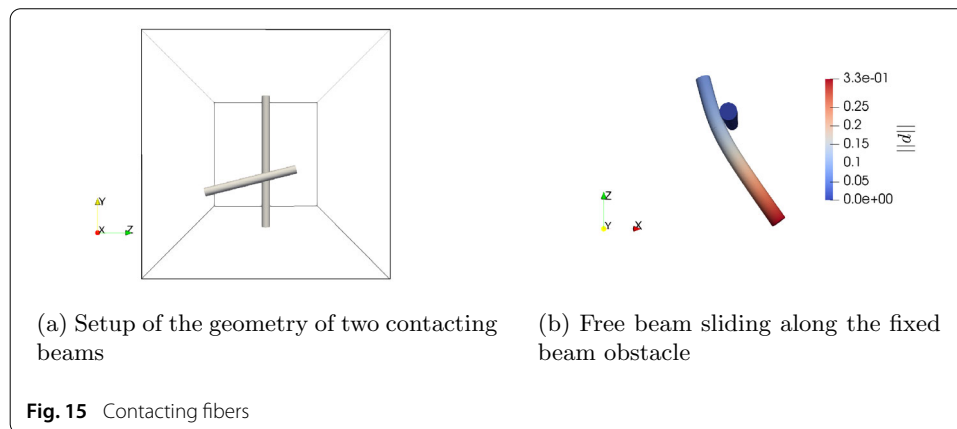
in channel direction, is prescribed at the channel inlet.

The behavior of an immersed fiber with a density $\rho = 1$, a length $l = 0.5$, a cross-sectional area $A = 0.166$, and accordingly a mass of $m = 0.0880$ is regarded. Note that the other material properties of the beam do not play a role in this example, since the fluid flow is set up to be constant along the beam length such that the results can be analyzed as an immersed mass in dependence on the penalty parameter. The time step size is chosen as $\Delta t = 0.01$.

In the limit case of freely movable light fibers, the fiber is expected to be transported by the fluid exactly at the velocity of the fluid. Figure 14a shows velocity results for the beam in dependence on the penalty parameter. As argued in Remark 7, within the fluid-to-beam one-way coupling variant, the penalty force is used to introduce a weak Dirichlet constraint into the structure problem. As in the case of the beam-to-fluid coupling one-way coupling variant, we thus expect linear convergence of the coupling violation towards zero for sufficiently large penalty parameters. Figure 14b exhibits this expected behavior. In particular, this validates the treatment of the geometry-dependence by a fixed-point iteration as proposed in Fig. instead of a full Newton method.

Contacting immersed beams

Next, we want to study the applicability of the proposed approach as a simplified method to capture the effect of fluid-driven contact of multiple beams. In the case of fluid-driven contact, generally, multiple constraints act on the beam, namely the non-penetration contact constraint in addition to the kinematic FBI constraint. In the case of the fluid-to-beam one-way coupling variant, where the fluid is assumed to not be affected by the beam, in general it is not possible for both constraints to be perfectly fulfilled at the same time.



Nevertheless, we will show the suitability of the proposed approach to model fluid-driven beam-to-beam contact under rather small violations of the one-way-coupling assumptions and moderate FBI penalty parameters, using a penalty contact method as introduced in [54].

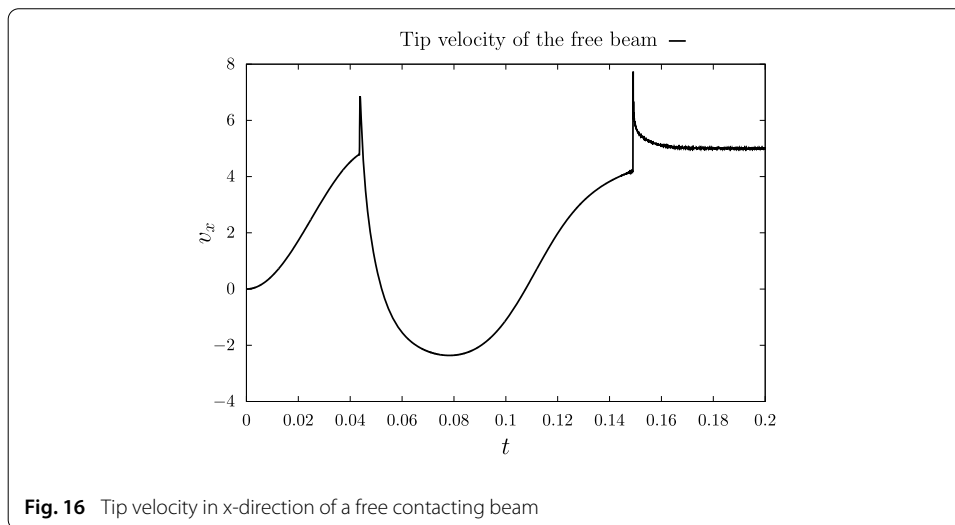
For this example, the fluid is again enclosed in a $1 \times 1 \times 3$ hexahedron, and the fluid at the inflow boundary is slowly accelerated to $v_x = 3$ along the longitudinal channel direction while using a time step size $\Delta t = 0.00005$. The first beam is secured at the middle of the bottom plane as in Fig. 7, but with a height of 0.9, a Young's modulus $E = 10^6$, a density $\rho = 10^3$, and a cross-section area $A = 0.25$. The second beam is freely flowing, has a Young's modulus of $E = 10^5$, a density $\rho = 10^{-3}$, a cross-sectional area $A = 0.25$, and is positioned at an angle of approximately 105° , 0.15 upstream from the first beam, as depicted in Fig. 15a. Contact itself is modeled using a tailored beam-to-beam contact formulation, as introduced in [54], with the contact penalty parameter 10^6 , while the FBI penalty parameter is chosen as 1.

The free flowing beam is expected to be transported with the fluid velocity until the time of first contact. After coming into contact, the free beam will begin sliding along one side of the fixed beam obstacle, as depicted in Fig. 15b, due to the geometrical unsymmetry of the problem setup.

Figure 16 shows the velocity of the tip of the free flowing beam, which is depicted as the right tip in Fig. 15a. The expected behavior of the sliding beam described above can be confirmed by the plotted velocity profile. This is to show that the proposed fluid-to-beam one-way coupling algorithm can adequately capture the behavior of contacting beams and is thus also extensible to practical applications moderately violating the one-way coupling assumptions.

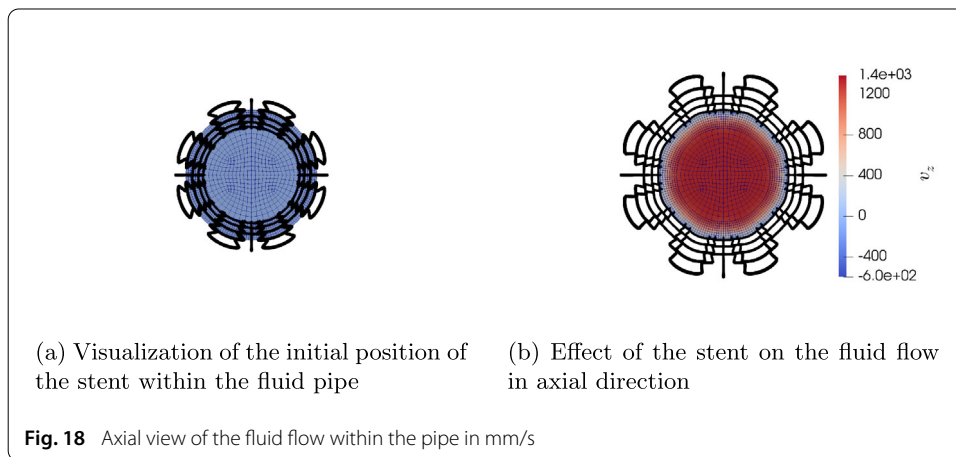
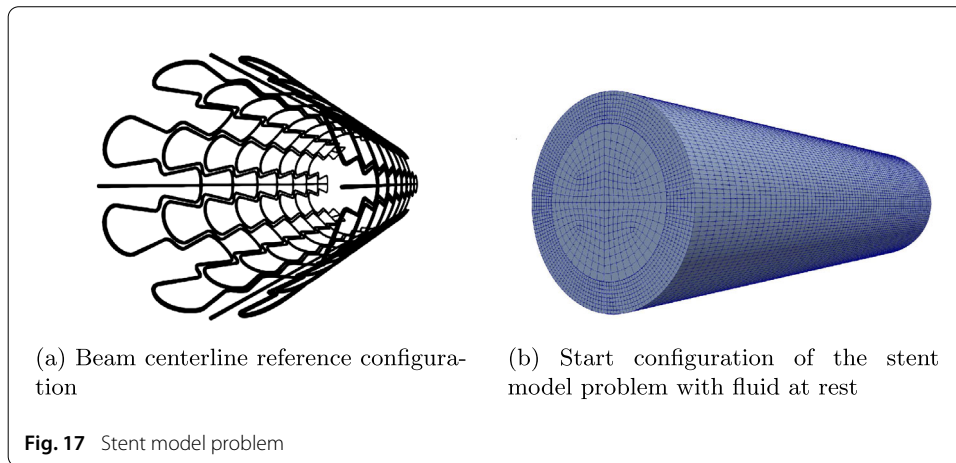
Immersed stent geometry

In "Introduction", the motivation of capturing phenomena related to stenting procedures through a bottom-up modeling approach was already mentioned. In the context of the direct interaction of blood flow and stent structure, two major complications determining the long-term outcome of the stenting procedure arise: firstly, the influence of the FSI force on the onset of stent migration. The complex phenomenon of stent migration depends on a delicate interplay of the blood-stent interaction, stent-vessel wall frictional contact and in the long term even on growth and remodeling of the vessel wall itself and can



thus not yet be reasonably captured by the proposed model. One of the involved factors, the growth and remodeling of the vessel wall, leads to the second major complication in the wake of stenting procedures: in-stent restenosis. Restenosis is currently thought to be connected to trauma of the vessel wall during the intervention itself, but also to a change in the flow pattern through the affected artery leading to oscillating wall shear stress (WSS) [55–57]. The additional roughness introduced by the stent itself has an effect on the latter. This motivates the following example of an immersed stent geometry inspired by material properties reported in [58] for the Taxus Liberté, which is used for stenting of coronary arteries. Nevertheless, since the beam constitutive model is given by a hyperelastic material model, not taking into account plasticity or pre-stressing, these aforementioned material properties have been adapted such that the fully expanded stent will touch the fluid boundary.

The stent geometry itself, depicted in Figure 17a, is based on [55], has an initial radius of 1.25 mm, a length of 13 mm, and is made up of 3488 Simo–Reissner beam elements. The beams have a Young’s modulus of $E = 6.2 \cdot 10^{10} \text{ g}/(\text{mm} \cdot \text{s}^2)$, a Poisson ratio $\nu = 0.3$, and a radius of $r = 0.03 \text{ mm}$. Figure 17 shows the setup of the model problem, for which the stent is half-way immersed in the fluid domain. The fluid domain consists of 113,800 fluid elements, has a radius of 1.75 mm, and a length of 15 mm. The density is set to $\rho = 0.001 \text{ g}/\text{mm}^3$, the dynamic viscosity $\gamma = 0.003 \text{ g}/(\text{mm} \cdot \text{s})$, and the time step size $\Delta t = 0.001 \text{ s}$ is used. The 113,800 fluid elements are further subdivided into two independent meshes: a coarser inner cylinder mesh with a radius of 1.25 mm and a finer outer layer as visualized in Fig. 17b. Both fluid meshes are then coupled using the mortar finite element method for surface coupling with condensed dual Lagrange multiplier shape functions as introduced in [59]. This mesh tying problem within the fluid field represents a rather complex application. This is to show that it is not only possible to include highly nonlinear phenomena within the structure field as demonstrated in “[Contacting immersed beams](#)”, but that the flexibility of the proposed approach also allows for complex models of the fluid field. For the application at hand, this is especially beneficial, since it is a priori known that the stent geometry will only move through the outer fluid layer, and it is expected that its effect on the fluid flow will also be restricted to this part of the fluid



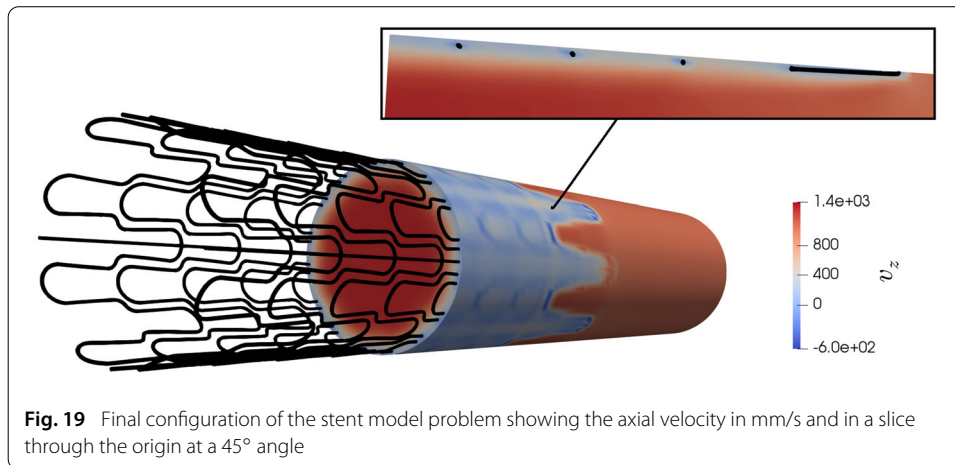
domain. Figure 18a shows the initial position of the stent at the edge of the outer fluid mesh layer.

To model the extension of the stent by the balloon, the internal pressure is successively increased from $t = 0$ s to $t = 0.1$ s using the formula

$$\begin{pmatrix} \bar{p} \cdot x \cdot t \\ \bar{p} \cdot y \cdot t \end{pmatrix}. \tag{30}$$

For a fully expanded stent geometry, as shown in Fig. 18b, $\bar{p} = 3.1 \cdot 10^3$ g/(mm · s²) is used. For simplicity, the fluid is assumed to be at rest and oblivious to the stent for the entire expansion procedure and the structure is simulated quasi-statically, thus neglecting its inertia. Once the stent is in place, the spatially constant fluid inflow velocity prescribed at the inlet boundary to the right is ramped up during 50 time steps to a maximum velocity of 1000 mm/s, while no penetration, free slip boundary conditions are enforced on the cylinder barrel of the fluid domain.

Figure 19 shows that the current method can capture the roughness introduced by the complex fully expanded stent geometry while taking into account material properties and expansion behavior of the structure. It is noteworthy that the exact location of stent wires/struts and, thus, the location of the imposed Dirichlet coupling conditions depends on the solution of the nonlinear beam problem modeling the stent and is thus not known a priori as in the other examples. To this end, Fig. 19 demonstrates that features of the



expanded stent geometry translate directly to the fluid flow. The fluid flow far from the stent is unaffected, as expected. On closer inspection of the slice through the origin at a 45° angle, backflow appears in the vicinity of the stent struts (colored in black), while the fluid is partly slowed down in between the cross beams. Also, the expansion procedure leads to larger displacements at the outskirts of the stent system, whereas the stabilizers and cross struts lead to stiffer a behavior far from the axial edges. This effect leads to even more flow back of the fluid near the upstream edge of the stent, while a boundary layer of decreased velocity is created downstream, where the stent geometry displays a smaller radial displacement.

Note that this model problem still represents a considerable simplification of the complex interactions present in a stented artery. In particular the neglect of any vessel wall and according boundary conditions poses an immense change in the dynamics of the system model. Nevertheless, the result shall serve as a proof of concept that the effect of established stent models, using one-dimensional beam elements as proposed in [12–14], can be captured by our FBI scheme. This represents a sound starting point for extensions to more complex models by including surface-coupled fluid-structure interaction [29,30] to model the influence of the vessel wall, as well as solid-beam interaction such as in [24] in order to model interactions of the stent with the vessel wall.

Conclusions

We have proposed an embedded finite element method to submerge one-dimensional beam elements in a three-dimensional fluid mesh using a GPTS-type coupling discretization. The introduced kinematic constraint is enforced via a penalty regularization directly on the beam centerline, thus making it a truly mixed-dimensional 1D–3D coupling problem as well as a highly efficient modeling strategy for immersed slender structures regarding computational complexity as well as mesh creation. This methodology was implemented for one-way coupling of the fluid to the beam and vice versa introducing all relevant components for a novel weak Dirichlet–Neumann partitioned algorithm. For both cases, the convergence of the resulting system solution with respect to the penalty parameter as well as the robustness of the segmentation algorithm to arbitrary positions within the background mesh was shown. The spatial convergence behavior of the fluid solution computed with the proposed 1D–3D coupling approach under uniform mesh refinement

as compared with a finely-resolved 3D CFD solution was shown and the approach, thus, validated. The general possibility of modeling highly complex nonlinear problems such as contact as well as mesh tying within the structure and/or fluid domain, and thus the flexibility of the proposed approach, was demonstrated by selected examples. In addition, limitations of the proposed one-way coupling strategies such as the exact capturing of interface phenomena have been pointed out. A simplified motivational example for possible practical application in form of an immersed stent geometry has been given as proof of concept for our modeling approach, and the additional nonlinearity introduced by the geometry change of the stent was well captured and visualized.

Obvious further research relates to the extension of the shown one-way coupling schemes to a stable two-way coupling algorithm by applying sophisticated convergence acceleration techniques to deal with well-known challenges in the context of the added mass effect. From a more application-centered view, the extension of the proposed method to include classical surface-based FSI, i.e. to model blood-vessel wall interactions in addition to the demonstrated blood-stent interaction, is an additional topic of interest.

Acknowledgements

Some sketches in this work have been created using the Adobe Illustrator plug-in LaTeX2AI (<https://github.com/steinbrecher/LaTeX2AI>). We acknowledge financial support by University of the Bundeswehr Munich.

Funding

Open Access funding enabled and organized by Projekt DEAL.

Declarations

Competing interests

The authors declare that they have no competing interests.

Author details

¹Institute for Mathematics and Computer-Based Simulation, University of the Bundeswehr Munich, Neubiberg, Germany,

²Data Science & Computing Lab, University of the Bundeswehr Munich, Neubiberg, Germany.

Received: 17 December 2021 Accepted: 30 May 2022

Published online: 21 June 2022

References

1. Gay M, Zhang L, Liu WK. Stent modeling using immersed finite element method. *Comput Method Appl M.* 2006;195(33–36):4358–70. <https://doi.org/10.1016/j.cma.2005.09.012>.
2. Mitsos AP, Kakalis NMP, Ventikos YP, Byrne JV. Haemodynamic simulation of aneurysm coiling in an anatomically accurate computational fluid dynamics model: Technical note. *Neuroradiology.* 2007;50(4):341–7. <https://doi.org/10.1007/s00234-007-0334-x>.
3. O'Connor J, Revell A. Dynamic interactions of multiple wall-mounted flexible flaps. *J Fluid Mech.* 2019;870:189–216. <https://doi.org/10.1017/jfm.2019.266>.
4. Wang M, Avital EJ, Bai X, Ji C, Xu D, Williams JJR, Munjiza A. Fluid-structure interaction of flexible submerged vegetation stems and kinetic turbine blades. *Comp Part Mech.* 2019;7(5):839–48. <https://doi.org/10.1007/s40571-019-00304-6>.
5. Favier J, Dauptain A, Basso D, Bottaro A. Passive separation control using a self-adaptive hairy coating. *J Fluid Mech.* 2009;627:451–83. <https://doi.org/10.1017/s0022112009006119>.
6. Kunze S, Brückner C. Control of vortex shedding on a circular cylinder using self-adaptive hairy-flaps. *Comptes Rendus Mécanique.* 2012;340(1–2):41–56. <https://doi.org/10.1016/j.crme.2011.11.009>.
7. Simo JC. A finite strain beam formulation. the three-dimensional dynamic problem. part i *Comput Method Appl M.* 1985;49(1):55–70. [https://doi.org/10.1016/0045-7825\(85\)90050-7](https://doi.org/10.1016/0045-7825(85)90050-7).
8. Reissner E. On one-dimensional finite-strain beam theory: The plane problem. *Journal of Applied Mathematics and Physics (ZAMP).* 1972;23(5):795–804. <https://doi.org/10.1007/bf01602645>.
9. Meier C, Popp A, Wall WA. An objective 3D large deformation finite element formulation for geometrically exact curved Kirchhoff rods. *Comput Method Appl M.* 2014;278:445–78. <https://doi.org/10.1016/j.cma.2014.05.017>.
10. Meier C, Popp A, Wall WA. A locking-free finite element formulation and reduced models for geometrically exact Kirchhoff rods. *Comput Method Appl M.* 2015;290:314–41. <https://doi.org/10.1016/j.cma.2015.02.029>.
11. Meier C, Grill MJ, Wall WA, Popp A Geometrically exact beam elements and smooth contact schemes for the modeling of fiber-based materials and structures. *Int. J. Solids Struct.* 2018;154:124–146. <https://doi.org/10.1016/j.ijsolstr.2017.07.020>. Multiscale Modelling of Fibrous and Textile Materials
12. Tambača J, Kosor M, Čanić S. MD, DP Mathematical modeling of vascular stents. *SIAM J Appl Math.* 2010;70(6):1922–52. <https://doi.org/10.1137/080722618>.

13. Tambača J, Čanić S, Paniagua D A novel approach to modeling coronary stents using a slender curved rod model: A comparison between fractured xience-like and palmaz-like stents. In: Fitzgibbon, W., Kuznetsov, Y.A., Neittaanmäki, P., Périaux, J., Pironneau, O. (eds.) *Computational Methods in Applied Sciences*, pp. 41–58. Springer, Dordrecht (2009). https://doi.org/10.1007/978-90-481-3239-3_5
14. Zunino P, Tambača J, Cutrì E, Čanić S, Formaggia L, Migliavacca F. Integrated stent models based on dimension reduction: Review and future perspectives. *Ann Biomed Eng.* 2015;44(2):604–17. <https://doi.org/10.1007/s10439-015-1459-4>.
15. Peskin CS. Flow patterns around heart valves: A numerical method. *J Comput Phys.* 1972;10(2):252–71. [https://doi.org/10.1016/0021-9991\(72\)90065-4](https://doi.org/10.1016/0021-9991(72)90065-4).
16. Peskin CS. The immersed boundary method. *Acta Numer.* 2002;11:479–517. <https://doi.org/10.1017/s0962492902000077>.
17. Kim Y, Peskin CS. Penalty immersed boundary method for an elastic boundary with mass. *Phys Fluids.* 2007;19(5):053103. <https://doi.org/10.1063/1.2734674>.
18. Rauch AD, Vuong A-T, Yoshihara L, Wall WA. A coupled approach for fluid saturated poroelastic media and immersed solids for modeling cell-tissue interactions. *Int J Numer Meth Biomed Engng.* 2018;34(11):3139. <https://doi.org/10.1002/cnm.3139>.
19. Hesch C, Gil AJ, Arranz Carreño A, Bonet J, Betsch P. A mortar approach for Fluid-Structure interaction problems: Immersed strategies for deformable and rigid bodies. *Comput Method Appl M.* 2014;278:853–82. <https://doi.org/10.1016/j.cma.2014.06.004>.
20. Baaijens FPT. A fictitious domain/mortar element method for fluid-structure interaction. *Int J Numer Meth Fluids.* 2001;35(7):743–61. [https://doi.org/10.1002/1097-0363\(20010415\)35:7<743:aid-flid109>3.0.co;2-a](https://doi.org/10.1002/1097-0363(20010415)35:7<743:aid-flid109>3.0.co;2-a).
21. Liu WK, Kim DW, Tang S. Mathematical foundations of the immersed finite element method. *Comput Mech.* 2005;39(3):211–22. <https://doi.org/10.1007/s00466-005-0018-5>.
22. Li L, Henshaw WD, Banks JW, Schwendeman DW, Main A. A stable partitioned FSI algorithm for incompressible flow and deforming beams. *J Comput Phys.* 2016;312:272–306. <https://doi.org/10.1016/j.jcp.2016.02.002>.
23. Huang DZ, Avery P, Farhat C. An embedded boundary approach for resolving the contribution of cable subsystems to fully coupled fluid-structure interaction. *Int J Numer Methods Eng.* 2020. <https://doi.org/10.1002/nme.6322>.
24. Steinbrecher I, Mayr M, Grill MJ, Kremheller J, Meier C, Popp A. A mortar-type finite element approach for embedding 1D beams into 3D solid volumes. *Comput Mech.* 2020;66(6):1377–98. <https://doi.org/10.1007/s00466-020-01907-0>.
25. Tschisgale S, Fröhlich J. An immersed boundary method for the fluid-structure interaction of slender flexible structures in viscous fluid. *J Comput Phys.* 2020;423: 109801. <https://doi.org/10.1016/j.jcp.2020.109801>.
26. Wu MCH, Muchowski H, Johnson E, Rajanna MR, Hsu M-C. Immersogeometric fluid-structure interaction modeling and simulation of transcatheter aortic valve replacement. *Comput Methods Appl Mech Eng.* 2019;357: 112556. <https://doi.org/10.1016/j.cma.2019.07.025>.
27. Kremheller J, Vuong A, Schrefler BA, Wall WA. An approach for vascular tumor growth based on a hybrid embedded/homogenized treatment of the vasculature within a multiphase porous medium model. *Int J Numer Meth Biomed Engng.* 2019;35(11):3253. <https://doi.org/10.1002/cnm.3253>. e3253 *cnm.3253*.
28. Crisfield MA, Jelenić G Objectivity of strain measures in the geometrically exact three-dimensional beam theory and its finite-element implementation. *Proc R Soc Lond A.* 1983;1999(455):1125–47. <https://doi.org/10.1098/rspa.1999.0352>.
29. Klöppel T, Popp A, Küttler U, Wall WA. Fluid-structure interaction for non-conforming interfaces based on a dual mortar formulation. *Comput Method Appl M.* 2011;200(45–46):3111–26. <https://doi.org/10.1016/j.cma.2011.06.006>.
30. Mayr M, Klöppel T, Wall WA, Gee MW. A temporal consistent monolithic approach to fluid-structure interaction enabling single field predictors. *SIAM J Sci Comput.* 2015;37(1):30–59. <https://doi.org/10.1137/140953253>.
31. Brezzi F, Bathe K-J. A discourse on the stability conditions for mixed finite element formulations. *Comput Method Appl M.* 1990;82(1–3):27–57. [https://doi.org/10.1016/0045-7825\(90\)90157-h](https://doi.org/10.1016/0045-7825(90)90157-h) Proceedings of the Workshop on Reliability in Computational Mechanics.
32. Béchet É, Moës N, Wohlmuth B. A stable Lagrange multiplier space for stiff interface conditions within the extended finite element method. *Int J Numer Meth Engng.* 2009;78(8):931–54. <https://doi.org/10.1002/nme.2515>.
33. Hautefeuille M, Annavarapu C, Dolbow JE. Robust imposition of Dirichlet boundary conditions on embedded surfaces. *Int J Numer Meth Engng.* 2011;90(1):40–64. <https://doi.org/10.1002/nme.3306>.
34. Sanders J, Puso MA. An embedded mesh method for treating overlapping finite element meshes. *Int J Numer Meth Engng.* 2012;91(3):289–305. <https://doi.org/10.1002/nme.4265>.
35. Kuchta M, Laurino F, Mardal K-A, Zunino P. Analysis and approximation of mixed-dimensional PDEs on 3D–1D domains coupled with Lagrange multipliers. *SIAM J Numer Anal.* 2021;59(1):558–82. <https://doi.org/10.1137/20M1329664>.
36. Kerfriden P, Claus S, Mihai I A mixed-dimensional CutFEM methodology for the simulation of fibre-reinforced composites. *Adv. Model. and Simul. in Eng. Sci.* 2020;7(1):1–26. <https://doi.org/10.1186/s40323-020-00154-5>
37. Jerison DS, Kenig CE. The Neumann problem on Lipschitz domains. *Bull Amer Math Soc.* 1981;4(2):203–8. <https://doi.org/10.1090/s0273-0979-1981-14884-9>.
38. Kuchta M, Mardal K, Mortensen M. Preconditioning trace coupled 3d–1d systems using fractional Laplacian. *Numer Methods Partial Differential Eq.* 2018;35(1):375–93. <https://doi.org/10.1002/num.22304>.
39. Luenberger, D.G., Ye, Y.: *Linear and Nonlinear Programming* vol. 1, 4th edn. Springer, (2016). <https://doi.org/10.1007/978-3-319-18842-3>
40. Hansbo P, Szepessy A. A velocity-pressure streamline diffusion finite element method for the incompressible navier-stokes equations. *Comput Method Appl M.* 1990;84(2):175–92. [https://doi.org/10.1016/0045-7825\(90\)90116-4](https://doi.org/10.1016/0045-7825(90)90116-4).
41. Franca LP, Frey SL Stabilized finite element methods: li. the incompressible navier-stokes equations. *Comput. Method. Appl. M.* 99(2–3), 209–233 (1992). [https://doi.org/10.1016/0045-7825\(92\)90041-h](https://doi.org/10.1016/0045-7825(92)90041-h)
42. Schott B, Rasthofer U, Gravemeier V, Wall WA. A face-oriented stabilized nitsche-type extended variational multiscale method for incompressible two-phase flow. *Int J Numer Meth Engng.* 2014;104(7):721–48. <https://doi.org/10.1002/nme.4789>.

43. Brüls O, Cardona A. On the use of lie group time integrators in multibody dynamics. *J Comput Nonlin Dyn*. 2010;5:1–13. <https://doi.org/10.1115/1.4001370>.
44. Brüls O, Cardona A, Arnold M. Lie group generalized- α time integration of constrained flexible multibody systems. *Mech Mach Theory*. 2012;48:121–37. <https://doi.org/10.1016/j.mechmachtheory.2011.07.017>.
45. Chung J, Hulbert GM. A time integration algorithm for structural dynamics with improved numerical dissipation: The generalized- α method. *J Appl Mech*. 1993;60(2):371–5. <https://doi.org/10.1115/1.2900803>.
46. Joosten MM, Dettmer WG, Perić D. On the temporal stability and accuracy of coupled problems with reference to fluid-structure interaction. *Int J Numer Meth Fluids*. 2010;64(10–12):1363–78. <https://doi.org/10.1002/flid.2333>.
47. Bukač M, Čanić S, Muha B, Glowinski R. An operator splitting approach to the solution of fluid-structure interaction problems in hemodynamics. In: Glowinski, R., Osher, S.J., Yin, W. (eds.) *Splitting Methods in Communication, Imaging, Science, and Engineering*, pp. 731–772. Springer, Cham (2016). https://doi.org/10.1007/978-3-319-41589-5_22
48. Causin P, Gerbeau JF, Nobile F. Added-mass effect in the design of partitioned algorithms for fluid-structure problems. *Comput Method Appl M*. 2005;194(42–44):4506–27. <https://doi.org/10.1016/j.cma.2004.12.005>.
49. Irons BM, Tuck RC. A version of the aitken accelerator for computer iteration. *Int J Numer Meth Engng*. 1969;1(3):275–7. <https://doi.org/10.1002/nme.1620010306>.
50. Küttler U, Wall WA. Fixed-point fluid-structure interaction solvers with dynamic relaxation. *Comput Mech*. 2008;43(1):61–72. <https://doi.org/10.1007/s00466-008-0255-5>.
51. Steinbrecher I, Popp A. MeshPy – A general purpose 3D beam finite element input generator. <https://compsim.gitlab.io/codes/meshpy>
52. BACI: A Comprehensive Multi-Physics Simulation Framework. <https://baci.pages.gitlab.lrz.de/website>
53. Laurino F, Zunino P. Derivation and analysis of coupled pdes on manifolds with high dimensionality gap arising from topological model reduction. *ESAIM: Math. Model. Numer. Anal.* 2019;53(6): 2047–2080 . <https://doi.org/10.1051/m2an/2019042>
54. Meier C, Popp A, Wall WA. A finite element approach for the line-to-line contact interaction of thin beams with arbitrary orientation. *Comput Method Appl M*. 2016;308:377–413. <https://doi.org/10.1016/j.cma.2016.05.012>.
55. Auricchio F, Conti M, Ferrara A, Morganti S, Reali A. Patient-specific finite element analysis of carotid artery stenting: A focus on vessel modeling. *Int J Numer Meth Biomed Engng*. 2012;29(6):645–64. <https://doi.org/10.1002/cnm.2511>.
56. Liang F, Takagi S, Himeno R, Liu H. Multi-scale modeling of the human cardiovascular system with applications to aortic valvular and arterial stenoses. *Med Biol Eng Comput*. 2009;47(7):743–55. <https://doi.org/10.1007/s11517-009-0449-9>.
57. Holzapfel GA, Stadler M, Gasser TC. Changes in the mechanical environment of stenotic arteries during interaction with stents: Computational assessment of parametric stent designs. *J Biomech Eng*. 2005;127(1):166–80. <https://doi.org/10.1115/1.1835362>.
58. Mortier P, Holzapfel GA, De Beule M, Van Loo D, Taeymans Y, Segers P, Verdonck P, Verhegghie B. A novel simulation strategy for stent insertion and deployment in curved coronary bifurcations: Comparison of three drug-eluting stents. *Ann Biomed Eng*. 2009;38(1):88–99. <https://doi.org/10.1007/s10439-009-9836-5>.
59. Ehrl A, Popp A, Gravemeier V, Wall WA. A dual mortar approach for mesh tying within a variational multiscale method for incompressible flow. *Int J Numer Meth Fluids*. 2014;76(1):1–27. <https://doi.org/10.1002/flid.3920>.

Publisher's Note

Springer Nature remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.

Submit your manuscript to a SpringerOpen[®] journal and benefit from:

- Convenient online submission
- Rigorous peer review
- Open access: articles freely available online
- High visibility within the field
- Retaining the copyright to your article

Submit your next manuscript at ► [springeropen.com](https://www.springeropen.com)
