On the Application of Total Traction Equilibrium in Topology Optimization of Fluid-Structure Interactions

Mohamed Abdelhamid · Aleksander Czekanski

Received: date / Accepted: date

Abstract This work investigates the different techniques of enforcing traction equilibrium in topology optimization of fluid-structure interaction problems. In literature, force coupling between the solid and the fluid domains is usually achieved using internal volume integrals of hydrostatic stresses. In this work, we extend the traction equilibrium condition to include the viscous stress components as well as the hydrostatic components in what is termed the total stress formulation. We also investigate the different force calculation techniques, and how the mixed finite element formulation results in two conflicting approaches; external surface vs. internal volume integrals. Lastly, a numerical experiment reveals the effects of the different force coupling techniques on the optimized designs with commentary on complexity of implementation and convergence behaviour.

Keywords fluid-structure interactions · porous media · topology optimization · traction equilibrium

1 Introduction

Fluid-structure interactions refer to problems where a deformable or a movable structure interacts with a flowing or a stationary fluid. Examples of fluid-structure interactions in everyday life are abundant; the flow of blood through flexible arteries, the deflection of an elastic airplane wing under aerodynamic loads, and the vibration of bridges and tall buildings under wind loads (Richter, 2017). A large body of research work has been dedicated to the numerical analysis of these interactions (Hou et al., 2012; Dowell and Hall, 2001), and most multiphysics simulation packages - commercial and open source - include some degree of capability for solving these problems. However, the application of numerical design tools such as topology optimization to these problems is still lagging behind. This is mainly due to the multidisciplinary nature of the combined topology optimization of fluid-structure interaction problem and its strong nonlinear behavior resulting in numerous stability and convergence issues.

Density-based topology optimization methods first appeared in the seminal work by Bendsoe (1989) which marked the start of modern topology optimization in its current form. In density-based methods, each discretization unit (e.g. finite element) is interpolated between material and void throughout the optimization process. Penalization of a parameter of interest is used to force the final solution towards a discrete 0/1 state (Abdelhamid and Czekanski, 2021). Later, topology optimization was extended to Stokes fluid flow in Borrvall and Petersson (2003) using a parameter to control permeability for design parametrization. In Gersborg-Hansen et al. (2005), the authors extended the work to the Navier-Stokes equations and used the analogy of a 2D channel flow with varying thickness for design parametrization. They also recognized, along with Evgrafov (2005), the similarity of this design parametrization to Brinkman equations (Brinkman, 1947) of fluid flow in porous media (Nield and Bejan, 2017, p. 15). This approach - later termed Brinkman penalization - became the de facto method for topology optimization.
of fluid flow without much regard to its physical interpretation.

The first application of topology optimization to fluid-structure interactions appeared in Maute et al. (2002). The authors used a segregated domain formulation to define the three-field fluid-structure interaction problem (i.e. fluid, solid, and mesh). In the segregated formulation, there is no overlap between the fluid and the solid computational domains thus only allowing for dry topology optimization (i.e. the fluid-structure interface is not optimized). This formulation was used in Maute and Allen (2004) and extended to mechanism design in adaptive wings in Maute and Reich (2006). In Stanford and Ifju (2009b,a), the authors utilized the three-field, segregated domain formulation in the design of wing skeletons of micro air vehicles using a two material formulation; wing skin or carbon fiber reinforcement. Although - technically speaking - this was a form of wet topology optimization, due to using a two material formulation there was no fluid porous elements, and hence from a force coupling perspective it’s similar to the dry topology optimization described in Maute et al. (2002). In Kreissl et al. (2010), the authors performed topology optimization on a flexible micro-fluidic device. However, they only considered the effect of structural deformations on the fluid flow and ignored the fluid forces on the structure. In Vu-Huu et al. (2018), a polytree-based adaptive polygonal finite element method was used for topology optimization of fluid-submerged breakwater barriers. However, they used a low fidelity model such that the initial fluid loading was used to calculate the forces without further updates based on the changes in the design domain. Hence the fluid forces were calculated on the fully solid structure and no porosity was incorporated in the fluid domain.

The first appearance of the unified domain formulation was in Yoon (2010) where the fluid and solid computational domains overlapped such that the whole design domain - including the fluid-structure interface - could be optimized, hence the term wet topology optimization. In Yoon (2010), the author extended the fluid-solid force coupling to the entire fluid domain including porous elements but only considered the hydrostatic component of the fluid stress tensor. The divergence theorem was utilized to transfer the integral from surface to volume. This step has a serious implication on the accuracy of the force coupling to be discussed in detail in Section 4. The same unified formulation and force coupling was later utilized in Yoon (2014b,a, 2017) by the same author. More recently, in Lundgaard et al. (2018), the authors revisited the density-based unified formulation with special emphasis on the multi-disciplinary interaction between the fluid and the solid and its effects on the optimized designs. They also investigated different fluid flow conditions and demonstrated the significance of careful selection of interpolation functions and parameters to ensure convergence. Nonetheless, the work was also limited to hydrostatic traction equilibrium with a volume integral.

The novelty in this work can be summarized in two points: (i) we extend the traction equilibrium condition to include the total stress tensor such that viscous forces are also considered, and (ii) we demonstrate the effect of the existence of sharp gradients in the fluid flow on the accuracy of the traction equilibrium condition. Both points are further demonstrated using numerical examples on the topology optimization of fluid-structure interaction problems.

The rest of the paper is organized as follows; in Section 2, we state the governing equations for the fluid-structure interaction problem. In Section 3, we derive the traction equilibrium condition using the total stress description. In Section 4, we investigate the inaccuracies in the traction equilibrium condition and describe the available different force calculation techniques. In Section 5, we demonstrate - with numerical examples on the topology optimization of fluid-structure interaction problems - the effect of using the total vs. hydrostatic traction equilibrium and using the different force calculation techniques on the optimized designs. Finally, we conclude the article in Section 6.

2 Governing Equations of the Fluid-Structure Interaction Problem

The finite element equations are derived using the Galerkin method of weighted residuals for both the fluid and the solid governing equations. Assumptions that are to be followed in this work concerning the Brinkman penalization concept are that the porous media consists of a volumetric mixture of fluid and solid phases, the solid phase is homogeneous and isotropic on the macroscopic scale, and all the pores are interconnected (Nield and Bejan, 2017, p. 4).

In our implementation of the unified domain formulation, the solid computational domain \( \Omega_s \) is completely contained within the fluid computational domain \( \Omega_f \), so the fluid domain spans the whole computational domain \( \Omega \). The design domain \( \Omega_d \) is fully contained within the solid domain. In \( \Omega \), a design variable per each finite element determines whether its pure fluid, pure solid, or inbetween. Only the design variables within the design domain are allowed to change. It’s possible to exclude the non-design pure solid regions - if any - from the
fluid domain, but more efforts have to be put in defining the boundary conditions, so we opted not to follow this approach here.

Although the general procedure of traction coupling in this work applies to fluid-structure interaction problems with large deformations, for simplicity a small displacement assumption is used in this work without any loss of generality. Conformal or matching discretization is assumed at the fluid-structure interface to ensure simple, yet accurate, force coupling. For non-conformal meshing, other mapping techniques can be used to enforce momentum conservation at the interface (Farhat et al., 1998).

For the fluid flow, the Navier-Stokes equations are used in their viscous, incompressible form. For the steady-state case, the strong form of the partial differential equations is as follows (Reddy and Gartling, 2010, p. 10):

$$\nabla \cdot \mathbf{v} = 0, \quad \rho_f (\nabla \cdot \mathbf{v}) = \nabla \cdot \boldsymbol{\sigma}_f + \mathbf{f}_f - \alpha(\rho) \mathbf{v}, \quad \mathbf{f}_f = -p \mathbf{I} + \mu [\nabla \mathbf{v} + (\nabla \mathbf{v})^T],$$

$$\alpha(\rho) = \alpha_{\max} + \frac{1 + p_{\alpha}}{1 - \rho + p_{\alpha}}.$$

where \( \mathbf{v} \) is the fluid velocity, \( \rho_f \) is the fluid density, \( \boldsymbol{\sigma}_f \) is the Cauchy fluid stress tensor, \( \mathbf{f}_f \) is the external fluid force, \( p \) is the hydrostatic pressure, and \( \mu \) is the fluid dynamic viscosity. \( \mathbf{f}_f \) is assumed zero and dropped in the remainder of this paper. The last term in Eq. 2 is the Brinkman penalization term added as a volume force such that \( \rho \) represents the solid density of the design element (i.e. 1 for pure solid, 0 for pure fluid, and porous solid in between), \( \alpha_{\max} \) and \( \alpha_{\min} \) are the Brinkman penalization upper and lower limits, and \( p_{\alpha} \) is the Brinkman penalization interpolation parameter that is dependent on the Reynolds number.

As for the structure, the Navier-Cauchy equations are used assuming linear elasticity under steady-state conditions. The strong form of the partial differential equations is as follows (Lai et al., 2010, p. 168):

$$\nabla \cdot \mathbf{d} = 0, \quad \mathbf{d} = C \epsilon,$$

$$\epsilon = \frac{1}{2} [\nabla \mathbf{d} + (\nabla \mathbf{d})^T],$$

$$E(\rho) = E_{\min} + (E_{\max} - E_{\min}) \rho^{p_E}.$$

where \( \mathbf{d} \) is the engineering strain tensor, \( \mathbf{C} \) is the linear elasticity tensor, \( \epsilon \) is the strain, and \( \mathbf{d} \) is the solid displacement. The elastic modulus is interpolated using the modified SIMP approach where \( E_{\min} \) and \( E_{\max} \) are the lower and upper limits on \( E \) and \( p_E \) is the penalization parameter. The essential boundary conditions are defined as follows:

$$\text{Fluid No-slip: } \mathbf{v} = 0 \quad \text{on } \Gamma_{\text{vo}}, \quad \text{(9)}$$

$$\text{Fluid Inlet: } \mathbf{v} = \mathbf{v}_{\text{in}} \quad \text{on } \Gamma_{\text{vin}}, \quad \text{(10)}$$

$$\text{Fluid Outlet: } p = 0 \quad \text{on } \Gamma_{\text{vout}}, \quad \text{(11)}$$

$$\text{Structural Prescribed Displacement: } \mathbf{d} = 0 \quad \text{on } \Gamma_{\text{dis}}, \quad \text{(12)}$$

Note that the fluid no-slip boundary condition is only defined on the external non-solid walls, on the solid walls this boundary condition is satisfied implicitly through the volume force (i.e. Brinkman Penalization) appended to the momentum balance in Eq. 2. The natural boundary condition in a fluid-structure interaction problem is defined as follows:

$$\sigma_f \cdot \mathbf{n}_f = \mathbf{n}_s \cdot \mathbf{n}_s \quad \text{on } \Gamma_{\text{FSI}}, \quad \text{(13)}$$

where \( \mathbf{n}_f \) and \( \mathbf{n}_s \) are the normals to the fluid and solid surfaces respectively. The fluid-structure interface \( \Gamma_{\text{FSI}} \) is only clearly identified in the absence of porous solids. In the topology optimization formulation, all solids are porous to some degree, hence \( \Gamma_{\text{FSI}} \) is taken as all edges of solid finite elements. In the next section, we discuss the details of the traction equilibrium condition described generally in Eq. 13.

3 Derivation of the Traction Equilibrium using the Total Stress Tensor

In literature, the traction equilibrium condition was derived to include the hydrostatic stress component only, which is suitable for some cases with negligible viscous stresses (cf. Yoon (2010, p. 601) and Lundgaard et al. (2018, p. 989)). In this section, we extend the traction equilibrium condition to include the viscous stress components as well. The forces applied from the fluid domain on the solid domain are calculated directly from Eq. 13 as follows:

$$\mathbf{f}_s = \int_{\Gamma_{\text{FSI}}} \sigma_f \cdot \mathbf{n}_f \, d\Gamma.$$

which can be put into the finite element form by multiplying by the velocity shape - and test - functions vector \( \Psi \) then discretizing:

$$\mathbf{F}_s^* = \int_{\Gamma_{\text{FSI}}} \Psi \left( \sigma_{ij}^f \mathbf{n}_f^j \right) \, d\Gamma.$$

where the solid/fluid designation is placed as a superscript whenever index notation is used; as in \( \mathbf{F}_s^* \) which designates the discretized solid force in the spatial direction \( i \). Also, an uppercase \( \mathbf{F} \) is used to describe forces
in the discretized finite element form as opposed to a lowercase \( f \) describing forces in the strong form. Eq. 15 can be detailed in the finite element form as follows:

\[
\mathbf{F}_x = \int_{-1}^{+1} \int_{-1}^{+1} \left[ -\Phi^T \mathbf{p} + 2\mu \frac{\partial \Psi^T}{\partial x} \mathbf{U} \right] n_x^f + \mu \left( \frac{\partial \Psi^T}{\partial y} \mathbf{U} + \frac{\partial \Psi^T}{\partial x} \mathbf{V} \right) n_y^f |\mathbf{J}| \, d\xi \, d\eta; \tag{16}\]

\[
\mathbf{F}_y = \int_{-1}^{+1} \int_{-1}^{+1} \mu \left( \frac{\partial \Psi^T}{\partial y} \mathbf{U} + \frac{\partial \Psi^T}{\partial x} \mathbf{V} \right) n_x^f + \left( -\Phi^T \mathbf{p} + 2\mu \frac{\partial \Psi^T}{\partial y} \mathbf{V} \right) n_y^f |\mathbf{J}| \, d\xi \, d\eta. \tag{17}\]

where \( x \) and \( y \) are the spatial coordinates, \( \Phi \) is the pressure shape functions vector, \( \mathbf{U} \) and \( \mathbf{V} \) are the discretized nodal velocities in the \( x \) and \( y \) directions, \( \mathbf{p} \) is the discretized nodal pressures, the normal vector components \( n_x^f \) and \( n_y^f \) can be calculated from simple geometry or from the shape functions (cf. Reddy and Gartling (2010, p. 184)), and \( |\mathbf{J}| \) designates the natural coordinate along the finite element edge of interest in 2D (or surface in 3D). Note that \( |\mathbf{J}| \) is the typical Jacobian determinant, however it is to be evaluated at the numerical integration points along the finite element edge of interest, hence some references add a surface designation to it.

In the case of porous media typical in topology optimization, the fluid-structure interface is no longer explicitly defined, rather it’s taken as all the edges of all the solid finite elements. Hence, in previous research works on topology optimization of fluid-structure interactions (and in works on fluid-structure interactions with porous media in general), the divergence theorem is typically used to transform the surface integral into a volume integral as follows (cf. Yoon (2010, p. 598) and (Lundgaard et al., 2018, p. 989)):

\[
\iint_{S} \left( \Psi \sigma_{ij}^{f} \right) n_{j}^{f} \, d\Gamma = \iiint_{V} \nabla \cdot \left( \Psi \sigma_{ij}^{f} \right) \, dV = \iiint_{V} \left( \frac{\partial \Psi}{\partial x_{j}} \sigma_{ij}^{f} + \Psi \frac{\partial \sigma_{ij}^{f}}{\partial x_{j}} \right) \, dV. \tag{18}\]

where \( x_{j} \) is the spatial coordinate in direction \( j \). By applying Eq. 18 to Eq. 15, the finite element form of the solid forces in 2D geometry can be obtained as follows:

\[
\mathbf{F}_x = \int_{-1}^{+1} \int_{-1}^{+1} \left[ -\Phi^T \mathbf{p} + 2\mu \frac{\partial \Psi^T}{\partial x} \mathbf{U} \right] + \mu \left( \frac{\partial \Psi^T}{\partial y} \mathbf{U} + \frac{\partial \Psi^T}{\partial x} \mathbf{V} \right) |\mathbf{J}| \, d\xi \, d\eta; \tag{19}\]

\[
\mathbf{F}_y = \int_{-1}^{+1} \int_{-1}^{+1} \mu \left( \frac{\partial \Psi^T}{\partial y} \mathbf{U} + \frac{\partial \Psi^T}{\partial x} \mathbf{V} \right) + \left( -\Phi^T \mathbf{p} + 2\mu \frac{\partial \Psi^T}{\partial y} \mathbf{V} \right) |\mathbf{J}| \, d\xi \, d\eta. \tag{20}\]

where \( \xi \) and \( \eta \) are the natural finite element coordinates in 2D geometry. The higher derivatives of the shape functions are derived in Appendix A.1. Note that by using the divergence theorem, the external surface integrals in Eqs. 16 and 17 are transformed to internal volume integrals in Eqs. 19 and 20. The distinction between external and internal formulations in investigated in detail in the following section.

4 Accuracy of the Traction Equilibrium in Porous Fluid-Structure Interactions

To illustrate the accuracy issues with the traction equilibrium derivation in Section 3, we first describe the setup of the numerical example detailed in Fig. 1. The beam in a channel analysis problem features a beam placed inside a fluid channel with its base \( \Gamma_{in} \) fixed to a ground structure. The top and bottom surfaces of the channel \( \Gamma_{out} \) have a no-slip condition applied. A uniform inlet velocity (i.e. plug flow) of 0.004 m/s in the \( x \) direction is applied at the left surface \( \Gamma_{in} \), and a zero pressure outlet is applied to the right surface \( \Gamma_{out} \). An oval structure is vertically centered in the channel and fixed at its internal square surface. Fluid density and dynamic viscosity are assumed 1 kg/m\(^3\) and 5e-4 Pa.s. The dimensions reported in Fig. 1 are in meters. The dotted lines are imaginary mesh lines from the fluid perspective, they are to be used as locations of force calculations in the following discussion.

In this work, we utilize a quadrilateral meshing which is regular and structured wherever possible. P2P1 finite elements (i.e. 9 velocity nodes and 4 pressure nodes) that satisfy the Ladyzhenskaya-Babuska-Brezzi condition (cf. Reddy and Gartling (2010, p. 176)) are used.
with a low Reynolds number to avoid using stabilization techniques that artificially - but not necessarily accurately - dampen the discontinuities.

This first numerical experiment is concerned with calculating the total traction forces according to Eqs. 16 and 17 (i.e. external surface integrals) at a number of lines, each from its two opposing sides. A line of zero thickness in a 2D fluid flow should induce no losses across it and the traction forces calculated from its two opposing sides should theoretically be equal. The upside and downside directions\(^1\) for each line are defined in Fig. 1 such that the upside is aligned with the arrow while the downside is opposite to it. The arrows indicate the normals to the lines which are pointing away from the domain for external boundaries but arbitrary for interior boundaries.

In this numerical experiment, we implement a pure fluid flow analysis to demonstrate that this inaccuracy in force calculation exists in pure fluid flow irrelevant of fluid-solid interactions and/or the inclusion of porosity. The solid/design domains are assumed void and a no-slip condition is applied to their wet boundaries. The mesh size reported in this section is the maximum allowable element size which is dominant over the meshing domain except near the oval shape and the skewed line 2. The pressure and velocity fields for this problem are presented in Fig. 2.

In the first set of results in Table 1, we focus on the total traction measurements at lines 1, 2, and 3 which are located in pure fluid away from geometric fea-

\(^1\)This terminology is borrowed from the commercial software COMSOL Multiphysics (COMSOL, 2020, p. 307)
tions that might introduce discontinuities in the flow. The errors reported are calculated as the difference between the upside and downside measurements divided by their mean while only considering the absolute value of all measurements. As can be noted, the absolute total traction in the $y$ direction is almost the same from the downside and upside directions for lines 1, 2, and 3. For the $x$ direction, line 1 has the same absolute traction values from both sides while a relatively small difference appears for line 2 and less for line 3. These differences diminish in value with mesh refinement, and they could be further reduced by symmetrizing and regularizing the mesh around each line.

In the second set of results in Fig. 3, we focus on lines close to geometric features that might introduce discontinuities in the fluid flow; namely lines 4 to 9. Errors in the $x$ direction demonstrate consistent improvement with mesh refinement for all lines in Fig. 3 except for a minor deviation for line 8. As for errors in the $y$ direction, lines 4, 5, partially 7, and partially 8 show consistent improvement with mesh refinement. The other lines (6 and 9) show an erratic behavior but the errors are very small compared to the other cases. This numerical experiment proves that in the existence of discontinuities in the fluid flow (even pure fluid), a relatively coarse mesh is not enough to correctly resolve the traction equilibrium condition. In the following, we discuss the reason behind these discrepancies.

In a mixed finite element formulation where the velocities and pressures are the unknown variables, compatibility is satisfied as the velocity and pressure fields are shared - hence continuous - across finite element boundaries (Bathe, 2014, p. 231). The components in the force formulation that contain first order derivatives of the primitive variables (velocities and pressures) are the reason behind these discrepancies as they are not continuous across finite element boundaries. Such discrepancies don’t appear when using force formulations that don’t include their first derivatives. To further investigate this observation, we take a look at the first derivatives of the primitive variables at different mesh densities at line 4 since this particular line has a clear correlation between total traction errors and mesh density (cf. Fig. 3a). From the results presented in Fig. 4, it is clear that with mesh refinement the errors of the first derivatives between the downside and upside directions are getting smaller. The first derivatives w.r.t. the $x$ direction are much higher with a clear trend as opposed to those w.r.t. the $y$ direction which have much smaller values and mostly outliers with no consistent behavior. This is expected from the location (cf. Fig. 1) and flow conditions (cf. Fig. 2) around line 4. It’s worth reiterating that the pressure first derivatives appear in

\[ \begin{array}{cccccccc}
\text{Mesh Size} & \text{Upside } x & \text{Downside } x & \text{Error } x & \text{Upside } y & \text{Downside } y & \text{Error } y \\
\hline
0.06 & 2.943 & 2.943 & \times 10^{-4} & 1.766 & 1.766 & \times 10^{-4} \\
0.05 & 2.948 & 2.948 & 0.00\% & 1.769 & 1.769 & 0.00\% \\
0.04 & 2.954 & 2.954 & 0.00\% & 1.772 & 1.772 & 0.00\% \\
0.03 & 2.961 & 2.961 & 0.00\% & 1.777 & 1.777 & 0.00\% \\
0.02 & 2.967 & 2.967 & 0.00\% & 1.781 & 1.781 & 0.00\% \\
0.01 & 2.974 & 2.974 & 0.00\% & 1.785 & 1.785 & 0.00\% \\
\hline
0.06 & 1.720 & 1.732 & 0.66\% & 1.770 & 1.770 & 0.00\% \\
0.05 & 1.721 & 1.736 & 0.89\% & 1.773 & 1.773 & 0.00\% \\
0.04 & 1.724 & 1.738 & 0.78\% & 1.776 & 1.776 & 0.00\% \\
0.03 & 1.732 & 1.735 & 0.18\% & 1.780 & 1.780 & 0.00\% \\
0.02 & 1.733 & 1.736 & 0.19\% & 1.784 & 1.784 & 0.00\% \\
0.01 & 1.738 & 1.739 & 0.04\% & 1.789 & 1.789 & 0.00\% \\
\hline
0.06 & 9.206 & 9.189 & 0.18\% & 5.933 & 5.932 & 0.00\% \\
0.05 & 9.234 & 9.229 & 0.06\% & 5.941 & 5.941 & 0.00\% \\
0.04 & 9.270 & 9.270 & 0.01\% & 5.951 & 5.951 & 0.01\% \\
0.03 & 9.325 & 9.322 & 0.03\% & 5.962 & 5.961 & 0.00\% \\
0.02 & 9.366 & 9.365 & 0.02\% & 5.971 & 5.971 & 0.00\% \\
0.01 & 9.408 & 9.407 & 0.01\% & 5.981 & 5.981 & 0.00\%
\end{array} \]

\[ Data at the end locations of line 4 have been removed from Fig. 4. At location $y = 0.4$ m, the right hand side (downside) has a zero first derivative due to its close proximity to the no-slip top surface of the beam resulting in a 200% error for all values. At location $y = 1.0$ m, both sides have extremely small first derivatives, hence their error is not representative. \]
the force calculations only when using a volume integral as in Eqs. 19 and 20.

Furthermore, through extracting the pressure force components only using the surface - not volume - integrals at line 1 to 9, the pressure forces were identical but opposite in sign from the downside and upside directions (results not included) which further corroborates our reasoning about the cause of the force discrepancies.
Fig. 4: Absolute percentage errors in the first derivatives of the primitive variables at line 4.
The next logical step is to check the force calculations where the first derivatives of the pressure are used; that is using volume integrals. In order to do this, we perform the second numerical experiment which uses the same setup of the first experiment while using solid media with zero porosity in the design domain $\Omega_d$ instead of void. The solid material properties used are a Poisson’s ratio $\nu$ of 0.3 and elastic modulus lower and upper limits of $E_{\text{min}} = 0.3 \text{ N/m}^2$ and $E_{\text{max}} = 3 \times 10^9 \text{ N/m}^2$. Since all design variables are assumed one within the design domain (i.e. zero porosity), the value of penalization parameter $p_E$ is irrelevant. This setup should give similar pressure and velocity fields to the ones in Fig. 2 with minor deviation in the pressure field as the fluid now diffuses through the design domain.

To understand the set of results to be extracted from the second numerical experiment, we must first discuss the different techniques for calculating forces on a porous media as presented in Fig. 5. For simplicity, consider a single porous finite element surrounded by pure fluid elements from three sides and an external boundary from the fourth side. The surface integrals in Figs. 5a and 5b are evaluated using Eqs. 16 and 17 while the volume integral in Fig. 5c is evaluated using Eqs. 19 and 20.

In the upside surface integral in Fig. 5a, the forces on the porous element are calculated through surface integrals applied on its boundaries using information from the surrounding elements. The bottom edge has no adjacent fluid element, hence the only possible solution is using data from the porous element itself. In the downside surface integral in Fig. 5b, the forces on the porous element are calculated using a surface integral on its boundaries using information from the porous element itself. Although the downside and upside surface integrals share the same primitive variables on the boundaries (hence continuity), the first derivatives of the velocities are not the same. In the volume integral in Fig. 5c, the forces on the porous element are calculated within its volume using information from the porous element itself. Hence, the downside surface integral and the volume integral are equivalent which is in agreement with the definition of the divergence theorem in Eq. 18.

An important question arises at this stage, which force calculation technique is accurate? Or if this is too big of a question to answer, which force calculation technique should be used? In pure solid, non-porous fluid-structure interaction problems, only one type of surface integration is available, hence this question doesn’t arise naturally. There are already benchmark experimental work for fluid-structure interaction problems but only using pure non-porous solid media (Hessenthaler et al., 2017), so empirical benchmarking is not an option. In the authors’ opinion, the lack of empirical data on force calculations of fluid-structure interaction problems with porous media is mainly due to the difficulty of producing a solid material with homogenous and isotropic porosity (cf. first paragraph in Section 2).

Building upon the different force techniques discussed above, we consider the results from the second numerical experiment presented in Fig. 6. The total traction is calculated on the beam (cf. Fig. 1). As a benchmark, we also include the forces calculated in the pure fluid-structure interaction case without the Brinkman porosity model. The upside and downside forces are calculated while including the Brinkman porosity model. It’s clear from Fig. 6 that the upside force technique aligns...
well with the benchmark result\textsuperscript{3}. As for the downside force technique, the error is more pronounced and decreases rather slowly with mesh refinement. This is particularly significant since the downside surface integral is equivalent to the volume integral typically used in fluid-structure interactions with porous media. An extensive mesh refinement is not always practical, especially considering the complexity of the topology optimization problem of fluid-structure interactions. Note that this discrepancy between the upside and downside values doesn’t arise away from sharp gradients near the fluid-structure interface. For instance, consider the small imaginary square inside the beam (cf. Fig. 1), the upside and downside forces on this square - extracted from the second numerical experiment - show excellent agreement as presented in Fig. 7.

In the next section, we present some numerical results on the topology optimization of fluid-structure in-

\textsuperscript{3}This is a testament to the validity of the Brinkman porosity model when the interpolation parameters $\alpha_{\text{max}}$ and $p_{\alpha}$ are properly selected. Unfortunately these parameters are sensitive to the problem and can’t be easily generalized (Lundgaard et al., 2018, p. 987).
teraction problems to showcase the effects of total vs. hydrostatic stress definitions as well as external surface vs. internal volume force calculation techniques.

5 Numerical Examples on the Topology Optimization of Fluid-Structure Interaction Problems

The third numerical example, a beam in a channel optimization problem, follows closely the original example by Yoon (2010, p. 610). It consists of a beam placed inside a fluid channel and fixed from the bottom surface to a ground structure. The objective function is to minimize compliance under a volume fraction constraint of 10%. Inside the design domain (light gray), a smaller beam (dark gray) is designated as a non-design domain to prevent trivial solutions. A fully-developed parabolic flow profile is applied at the inlet boundary \( \Gamma_{v_{\text{in}}} \) with a maximum velocity of \( 1.0 \times 10^{-4} \) m/s. A zero pressure boundary condition is applied at the outlet boundary \( \Gamma_{v_{\text{out}}} \). A no-slip boundary condition is applied at the other external boundaries \( \Gamma_{v_{0}} \). The material properties are as follows: \( \rho_f = 1000 \text{ kg/m}^3 \), \( \mu = 0.001 \text{ kg m}^{-1} \text{s}^{-1} \), \( E_{\text{min}} = 0.3 \text{ N/m}^2 \), \( E_{\text{max}} = 3 \times 10^9 \text{ N/m}^2 \), and \( \nu = 0.3 \). These conditions give a Reynolds number of \( Re = \rho_f \cdot U_{\text{max}} L/\mu = 1000-1.0 \times 10^{-4} \cdot 400 \times 10^{-6} /0.001 = 0.04^4 \). A mesh size of 1 \( \mu \)m is used such that the total number of finite elements in the whole domain is 40,000.

The topology optimization problem is solved four times spanning the different combinations of possible traction equilibrium conditions. As a benchmark, the first result in Fig. 9a for hydrostatic stress evaluated using internal volume integrals used the same force calculation technique as that of Yoon (2010, p. 613)’s. The two designs bear some similarities, albeit our design doesn’t exhibit the same symmetry around the \( y \) axis. Our optimized objective function is \( 4.1014 \times 10^{-20} \) J vs. \( 3.4025 \times 10^{-20} \) J in Yoon (2010). Two reasons might have caused this minor discrepancy: (i) Yoon (2010) used a different Brinkman penalization interpolation formula in the form of \( \alpha(\rho) = \alpha_{\text{max}} - \rho \rho_{\alpha} \) as opposed to Eq. 4 used in this work, and (ii) although Yoon (2010) used a continuation scheme to raise the elastic modulus penalization factor to force a discrete solution, the details of this continuation scheme are unclear.

As mentioned earlier, we used quadratic Lagrange finite elements for the solid domain to ensure conformal mesh with the fluid domain. This type of finite elements, although less prone to checkerboarding than linear elements, can still exhibit checkerboarding at a high penalization factor. We opted not to employ filters so as to isolate the effects of the traction equilibrium condition from other factors.

In terms of the optimized designs, the hydrostatic stress formulation with internal volume integrals (Fig. 9a) is the simplest approach to implement with the best convergence behavior. In some cases when the viscous stresses are significant and cannot be ignored, implementing the total stress formulation becomes a necessity. At such a low Reynolds number as in the third numerical example, the total stress formulation with internal volume integrals (Fig. 9b) produced a different design from the hydrostatic formulation at a slight increase in the implementation complexity as can be noted in Eqs. 19 and 20. The main difficulty arises when implementing external surface integrals as there would be a need to create a connectivity matrix to link each edge of each finite element to its corresponding external

This Reynolds number was incorrectly reported in (Yoon, 2010, p. 613) as \( Re = 0.004 \). The characteristic length was not reported explicitly in Yoon (2010), but correlating with the other Reynolds numbers calculated for the same problem concluded it was \( L_c = 400 \mu \text{m} \).

Fig. 8: The third numerical experiment; the beam in a channel optimization problem.
Fig. 9: Optimized designs of the beam in a channel optimization problem for different traction equilibrium conditions.
one. In addition, convergence deteriorates significantly when combining external surface integrals with total stress formulation (Fig. 9d) as opposed to hydrostatic stress formulation (Fig. 9c). It appears that, between iterations, the fluctuations in the pressure derivatives is not as severe as those in the velocities derivatives judging from the drastically different design produced in Fig. 9d.

6 Conclusions

Topology optimization of fluid-structure interaction problems is an emerging field in numerical design of multiphysics systems. Due to the complexity and the inherent non-linearity, the published work on this area has been limited so far. In this work, we focus on the different techniques for traction equilibrium between the solid and fluid domains.

We extended the traction equilibrium condition to use the total stress formulation as opposed to the hydrostatic stress formulation used in previous literature. The viscous stress components become more pronounced for low Reynolds numbers, hence total stress formulation (Fig. 9d) as opposed to hydrostatic stress formulation used in previous literature. At the other hand, external surface integrals are easier to implement and more stable w.r.t. internal volume integrals. At one hand, internal volume integrals are the de facto force calculation method in pure fluid-structure interaction problems, but they are more difficult to implement and cause more convergence issues. Numerical stabilization of these convergence issues might be the reason behind the drastically different design observed in the third numerical experiment.

Appendix

A.1 Derivation of the Higher Derivatives of the Shape Functions

In analyzing fluid flow using the finite element method, two main reasons may necessitate computing the higher derivatives of the shape functions: (i) using Galerkin/Least-Squares stabilization with higher order elements (Reddy and Gartling, 2010, p. 202), and (ii) applying the divergence theorem on the fluid-solid force coupling while considering the viscous stresses. In the following derivation, we assume a linear geometric order such that the global spatial coordinates are $x = \tilde{\Psi}^T x$ and $y = \tilde{\Psi}^T y$ where $\tilde{\Psi}$ is a vector of linear shape functions with $x$ and $y$ being the linear nodal coordinates corresponding to $\tilde{\Psi}$. The vector of velocity shape functions $\Psi$ can be linear or higher order. Starting from the Jacobian with $\psi_i$ as the $i^{th}$ component of $\psi$ (Bathe, 2014, p. 346):

$$\begin{align*}
\left\{ \frac{\partial \psi_i}{\partial \xi} \right\} &= \left[ J \right] \left\{ \frac{\partial \psi_i}{\partial x} \right\} = \left[ \frac{\partial \xi}{\partial \xi} \frac{\partial \eta}{\partial \eta} \frac{\partial \xi}{\partial \xi} \frac{\partial \eta}{\partial \eta} \frac{\partial \xi}{\partial \xi} \frac{\partial \eta}{\partial \eta} \right] \left\{ \frac{\partial \psi_i}{\partial \xi} \right\} ,
\end{align*}$$

By taking the derivative of Eq. 21 w.r.t. $x$ and $y$ separately:

$$\begin{align*}
\left\{ \frac{\partial^2 \psi_i}{\partial \xi \partial \eta} \right\} &= \left\{ \frac{\partial^2 \psi_i}{\partial \xi \partial \eta} \right\} + \left\{ \frac{\partial^2 \psi_i}{\partial \xi \partial \eta} \right\} ,
\left\{ \frac{\partial^2 \psi_i}{\partial \xi \partial \eta} \right\} &= \left\{ \frac{\partial^2 \psi_i}{\partial \xi \partial \eta} \right\} + \left\{ \frac{\partial^2 \psi_i}{\partial \xi \partial \eta} \right\} ,
\end{align*}$$

Since $x$ and $y$ are independent:

$$\begin{align*}
\frac{\partial^2 y}{\partial \eta \partial \eta} &= 0, \frac{\partial^2 x}{\partial \eta \partial \eta} = 0, \frac{\partial^2 y}{\partial \xi \partial \eta} = 0, \frac{\partial^2 y}{\partial \xi \partial \eta} = 0. \quad (24)
\end{align*}$$

In order to calculate the remaining non-zero higher order derivatives in the first terms of the RHS of Eqs. 22 and 23, we use the multivariate chain rule. Recall that $\partial \psi_i / \partial \xi$ - for instance - is generally not equal to $\partial \psi_i / \partial x$ for multivariate functions. Instead the following applies:

$$\begin{align*}
\frac{\partial^2 x}{\partial \xi \partial \xi} &= \frac{\partial^2 x}{\partial \xi \partial \xi} \frac{\partial \xi}{\partial \xi} + \frac{\partial^2 x}{\partial \xi \partial \xi} \frac{\partial \xi}{\partial \xi} ,
\frac{\partial^2 y}{\partial \xi \partial \xi} &= \frac{\partial^2 y}{\partial \xi \partial \xi} \frac{\partial \xi}{\partial \xi} + \frac{\partial^2 y}{\partial \xi \partial \xi} \frac{\partial \xi}{\partial \xi} ,
\frac{\partial^2 y}{\partial \xi \partial \eta} &= \frac{\partial^2 y}{\partial \xi \partial \xi} \frac{\partial \xi}{\partial \xi} + \frac{\partial^2 y}{\partial \xi \partial \eta} \frac{\partial \xi}{\partial \eta} ,
\frac{\partial^2 y}{\partial \xi \partial \eta} &= \frac{\partial^2 y}{\partial \xi \partial \xi} \frac{\partial \xi}{\partial \xi} + \frac{\partial^2 y}{\partial \xi \partial \eta} \frac{\partial \xi}{\partial \eta} ,
\frac{\partial^2 y}{\partial \xi \partial \eta} &= \frac{\partial^2 y}{\partial \xi \partial \xi} \frac{\partial \xi}{\partial \xi} + \frac{\partial^2 y}{\partial \xi \partial \eta} \frac{\partial \xi}{\partial \eta} ,
\end{align*}$$

Due to the linear geometric order assumption:

$$\begin{align*}
\frac{\partial^2 x}{\partial \xi \partial \xi} &= 0, \frac{\partial^2 x}{\partial \eta \partial \eta} = 0, \frac{\partial^2 y}{\partial \xi \partial \xi} = 0, \frac{\partial^2 y}{\partial \eta \partial \eta} = 0. \quad (25)
\end{align*}$$

The remaining first order terms in the RHS of Eqs. 25 can be calculated by inverting the Jacobian as follows:

$$\begin{align*}
\left[ \frac{\partial \xi}{\partial \xi} \frac{\partial \eta}{\partial \eta} \frac{\partial \xi}{\partial \xi} \frac{\partial \eta}{\partial \eta} \frac{\partial \xi}{\partial \xi} \frac{\partial \eta}{\partial \eta} \right]^{-1} &= \left[ \frac{\partial \xi}{\partial \xi} \frac{\partial \eta}{\partial \eta} \frac{\partial \xi}{\partial \xi} \frac{\partial \eta}{\partial \eta} \frac{\partial \xi}{\partial \xi} \frac{\partial \eta}{\partial \eta} \right].
\end{align*}$$

(27)
By rearranging Eqs. 22 and 23 and using the above information:

\[
\begin{align*}
\frac{\partial^2 \psi}{\partial x^2} \frac{\partial x}{\partial \xi} &+ \frac{\partial \psi}{\partial \eta} \frac{\partial x}{\partial \eta} = 0 \\
\frac{\partial^2 \psi}{\partial y^2} \frac{\partial y}{\partial \xi} &+ \frac{\partial \psi}{\partial \eta} \frac{\partial y}{\partial \eta} = 0 \\
\frac{\partial^2 \psi}{\partial \eta^2} \frac{\partial \eta}{\partial \xi} &+ \frac{\partial \psi}{\partial \eta} \frac{\partial \eta}{\partial \eta} = 0 \\
\end{align*}
\]

\[ (28) \]

Rearranging again:

\[
\begin{align*}
\left[ \begin{array}{cc}
\frac{\partial^2 \psi}{\partial x^2} & \frac{\partial \psi}{\partial \eta} \\
\frac{\partial \psi}{\partial \eta} & \frac{\partial^2 \psi}{\partial y^2}
\end{array} \right] 
\left[ \begin{array}{c}
\frac{\partial x}{\partial \xi} \\
\frac{\partial y}{\partial \eta}
\end{array} \right] &= 
\left[ \begin{array}{c}
\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} \\
\frac{\partial^2 \psi}{\partial \eta^2}
\end{array} \right] 
\left[ \begin{array}{c}
\frac{\partial x}{\partial \xi} \\
\frac{\partial y}{\partial \eta}
\end{array} \right] \\

\frac{\partial^2 \psi}{\partial \eta^2} &= 
\left[ \begin{array}{c}
\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} \\
\frac{\partial^2 \psi}{\partial \eta^2}
\end{array} \right] 
\left[ \begin{array}{c}
\frac{\partial x}{\partial \xi} \\
\frac{\partial y}{\partial \eta}
\end{array} \right] \\
\end{align*}
\]

\[ (30) \]

By solving these two systems of equations, the higher derivatives of the shape functions w.r.t. the global coordinates \(x\) and \(y\) can be obtained.

**Replication of results** The authors have included all dimensions and material parameters needed to replicate the results.

**Declarations**

**Conflict of interest** The authors declare they have no conflict of interest.

**Funding** There is no funding source.

**Ethical approval** This article does not contain any studies with human participants or animals performed by any of the authors.

**References**

Abdelhamid M, Czekanski A (2021) Revisiting non-convexity in topology optimization of compliance minimization problems. Engineering Computations 39(3):893–915, DOI 10.1108/EC-01-2021-0052

Bathe KJ (2014) Finite Element Procedures. Klaus-Jurgen Bathe

Bendsoe MP (1989) Optimal shape design as a material distribution problem. Structural Optimization 1(4):193–202, DOI 10.1007/BF01650949

Borrvall T, Petersson J (2003) Topology optimization of fluids in Stokes flow. International Journal for Numerical Methods in Fluids 41(1):77–107, DOI 10.1002/fld.426

Brinkman HC (1947) A calculation of the viscous force exerted by a flowing fluid on a dense swarm of particles. Applied Scientific Research, Section A 1(1):27–34, DOI 10.1007/BF02120313

COMSOL (2020) COMSOL Multiphysics Reference Manual 5.6

Dowell EH, Hall KC (2001) Modeling of Fluid-Structure Interaction. Annual Review of Fluid Mechanics 33(1):445–490, DOI 10.1146/annurev.fluid.33.1.445

Evgrafov A (2005) The Limits of Porous Materials in the Topology Optimization of Stokes Flows. Applied Mathematics and Optimization 52(3):263–277, DOI 10.1007/s00245-005-0828-z

Farhat C, Lesoinne M, Le Tallec P (1998) Load and motion transfer algorithms for fluid/structure interaction problems with non-matching discrete interfaces: Momentum and energy conservation, optimal discretization and application to aeroelasticity. Computer Methods in Applied Mechanics and Engineering 157(1-2):95–114, DOI 10.1016/S0045-7825(97)00216-8

Gersborg-Hansen A, Sigmund O, Haber R (2005) Topology optimization of channel flow problems. Structural and Multidisciplinary Optimization 30(3):181–192, DOI 10.1007/s00158-004-0508-7

Hessenthaler A, Gaddum NR, Holub O, Sinkus R, Röhrle O, Nordsetten D (2017) Experiment for validation of fluid-structure interaction models and algorithms. International Journal for Numerical Methods in Biomedical Engineering 33(9):1–13, DOI 10.1002/cnm.2848

Hou G, Wang J, Layton A (2012) Numerical Methods for Fluid-Structure Interaction — A Review. Communications in Computational Physics 12(2):337–377, DOI 10.4208/cicp.291210.290411s

Kreissl S, Pingen G, Evgrafov A, Maute K (2010) Topology optimization of flexible micro-fluidic devices. Structural and Multidisciplinary Optimization 42(4):495–516, DOI 10.1007/s00158-010-0526-6

Lai WM, Rubin D, Krempl E (2010) Introduction to Continuum Mechanics. Elsevier, New York, DOI 10.1016/B978-0-7506-8560-3.50001-2

Lundgaard C, Alexandersen J, Zhou M, Andreasen CS, Sigmund O (2018) Revisiting density-based topology optimization for fluid-structure-interaction problems. Structural and Multidisciplinary Optimization
On the Application of Total Traction Equilibrium in Topology Optimization of Fluid-Structure Interactions

58(3):969–995, DOI 10.1007/s00158-018-1940-4

Maute K, Allen M (2004) Conceptual design of aeroelastic structures by topology optimization. Structural and Multidisciplinary Optimization 27(1-2):27–42, DOI 10.1007/s00158-003-0362-z

Maute K, Reich G (2006) Integrated Multidisciplinary Topology Optimization Approach to Adaptive Wing Design. Journal of Aircraft 43(1):253–263, DOI 10.2514/1.12802

Maute K, Nikbay M, Farhat C (2002) Conceptual Layout of Aeroelastic Wing Structures by Topology Optimization. In: 43rd AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference, American Institute of Aeronautics and Astronautics, Reston, Virginia, pp 1–11, DOI 10.2514/6.2002-1480

Nield DA, Bejan A (2017) Convection in Porous Media, 5th edn. Springer International Publishing, Cham, DOI 10.1007/978-3-319-49562-0

Reddy JN, Gartling DK (2010) The Finite Element Method in Heat Transfer and Fluid Dynamics. CRC Press

Richter T (2017) Fluid-Structure Interactions - Models, Analysis, and Finite Elements, Lecture Notes in Computational Science and Engineering, vol 118. Springer International Publishing, Cham, DOI 10.1007/978-3-319-63970-3

Stanford B, Ifju P (2009a) Aeroelastic topology optimization of membrane structures for micro air vehicles. Structural and Multidisciplinary Optimization 38(3):301–316, DOI 10.1007/s00158-008-0292-x

Stanford B, Ifju P (2009b) Multi-Objective Topology Optimization of Wing Skeletons for Aeroelastic Membrane Structures. International Journal of Micro Air Vehicles 1(1):51–69, DOI 10.1260/1756-8293.1.1.51

Vu-Huu T, Phung-Van P, Nguyen-Xuan H, Abdel Wabeh M (2018) A polytree-based adaptive polygonal finite element method for topology optimization of fluid-submerged breakwater interaction. Computers & Mathematics with Applications 76(5):1198–1218, DOI 10.1016/j.camwa.2018.06.008

Yoon GH (2010) Topology optimization for stationary fluid-structure interaction problems using a new monolithic formulation. International Journal for Numerical Methods in Engineering 82(5):591–616, DOI 10.1002/nme.2777

Yoon GH (2014a) Compliant topology optimization for planar passive flap micro valve. Journal of nanoscience and nanotechnology 14(10):7585–91, DOI 10.1166/jnn.2014.9552

Yoon GH (2014b) Stress-based topology optimization method for steady-state fluid-structure interaction problems. Computer Methods in Applied Mechanics and Engineering 278:499–523, DOI 10.1016/j.cma.2014.05.021

Yoon GH (2017) Brittle and ductile failure constraints of stress-based topology optimization method for fluid–structure interactions. Computers & Mathematics with Applications 74(3):398–419, DOI 10.1016/j.camwa.2017.04.015