Journal of Computational Physics 261 (2014) 162–171

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

# A tangential regularization method for backflow stabilization in hemodynamics

Cristóbal Bertoglio [a,*], Alfonso Caiazzo [b]

[a] Institute for Computational Mechanics, Technische Universität München, 85747 Garching b. München, Germany
[b] Weierstrass Institute for Applied Analysis and Stochastics, 10117 Berlin, Germany

ARTICLE INFO

*Article history:*
Received 17 June 2013
Received in revised form 24 October 2013
Accepted 29 December 2013
Available online 6 January 2014

*Keywords:*
Navier–Stokes equations
Backflow stabilization
Blood flow
Finite element method

ABSTRACT

In computational simulations of fluid flows, instabilities at the Neumann boundaries may appear during backflow regime. It is widely accepted that this is due to the incoming energy at the boundary from the convection term, which cannot be controlled when the velocity field is unknown. Hence, we propose a stabilized formulation based on a local regularization of the fluid velocity along the tangential directions on the Neumann boundaries. The stabilization term is proportional to the amount of backflow, and does not require any further assumption on the velocity profile. The performance of the method is assessed on a two- and three-dimensional Womersley flows, as well as considering a hemodynamic physiological regime in a patient-specific aortic geometry.



## 1. Introduction

Let us consider an incompressible fluid in a domain $\Omega \subset \mathbb{R}^d$, $d = 2, 3$, whose boundary is decomposed as

$$\partial\Omega := \Gamma_{\text{in}} \cup \Gamma_{\text{out}} \cup \Sigma,$$

with $\Gamma_{\text{in}}$ and $\Gamma_{\text{out}}$ denoting the boundaries with Dirichlet data (i.e., given velocity profile) and Neumann data (i.e., given stresses), respectively. We consider an incompressible, Newtonian fluid, modeled through the incompressible Navier–Stokes equations for the velocity $\boldsymbol{u} : \Omega \times \mathbb{R}^+ \to \mathbb{R}^d$ and the pressure $p : \Omega \times \mathbb{R}^+ \to \mathbb{R}$:

$$\begin{cases} \rho\partial_t \boldsymbol{u} + \rho\boldsymbol{u} \cdot \nabla\boldsymbol{u} - \nabla \cdot \boldsymbol{\sigma}(\boldsymbol{u}, p) = \boldsymbol{0} & \text{in } \Omega, \\ \nabla \cdot \boldsymbol{u} = 0 & \text{in } \Omega, \\ \boldsymbol{u} = \boldsymbol{u}_{\text{in}} & \text{on } \Gamma_{\text{in}}, \\ \boldsymbol{u} = \boldsymbol{0} & \text{on } \Sigma, \\ \boldsymbol{\sigma}(\boldsymbol{u}, p)\boldsymbol{n} = -p_{\text{out}}\boldsymbol{n} & \text{on } \Gamma_{\text{out}}. \end{cases} \tag{1}$$

In (1), $\rho$ stands for the fluid density, $\mu$ denotes the dynamic fluid viscosity and the fluid Cauchy-stress tensor is given by $\boldsymbol{\sigma}(\boldsymbol{u}, p) := -p\boldsymbol{I} + 2\mu\boldsymbol{\epsilon}(\boldsymbol{u})$ and $\boldsymbol{\epsilon}(\boldsymbol{u}) := (\nabla\boldsymbol{u} + \nabla\boldsymbol{u}^{\mathrm{T}})/2$. Furthermore, $\boldsymbol{u}_{\text{in}}$ represents a given velocity profile and $p_{\text{out}}$ a given pressure data.

Let us denote with $(\cdot,\cdot)_X$ the usual scalar product in the Sobolev space $L^2(X)$, for $X \subset \mathbb{R}^d$, and with $\|\cdot\|_{0,X}$ the associated norm. Then, the quantities

* Correspondence to: C. Bertoglio, Lehrstuhl für Numerische Mechanik, Technische Universität München, Boltzmannstr. 15, 85747 Garching b. München, Germany.

*E-mail addresses:* bertoglio@lnm.mw.tum.de (C. Bertoglio), caiazzo@wias-berlin.de (A. Caiazzo).

0021-9991/$ – see front matter 
http://dx.doi.org/10.1016/j.jcp.2013.12.057

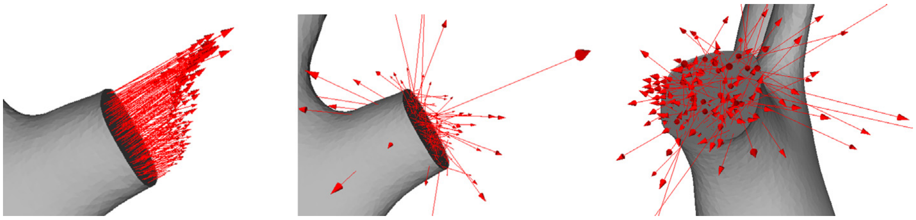

**Fig. 1.** A typical backflow instability arising in blood flow simulations: velocity vectors on the outlets at the time of peak outflow (left) and the time when backflow starts in the first outlet (center and right).

$$E(t) := \frac{\rho}{2}\|\boldsymbol{u}\|^2_{0,\Omega}, \qquad D(t) := 2\mu\big\|\boldsymbol{\epsilon}\big(\boldsymbol{u}(s)\big)\big\|^2_{0,\Omega}$$

denote the total (purely kinetic) energy of the 3D fluid system given by (1) and the dissipative effects, respectively. Using standard arguments, in the case of an isolated system, i.e., $\boldsymbol{u}_{\text{in}} = \boldsymbol{0}$ and $p_{\text{out}} = 0$, the energy balance of system (1) yields

$$\frac{d}{dt}E(t) = -D(t) - \left(\frac{\rho}{2}|\boldsymbol{u}|^2, \boldsymbol{u}\cdot\boldsymbol{n}\right)_{\Gamma_{\text{out}}}. \tag{2}$$

Notice that the last term of the right hand side of (2) cannot be bounded, when the velocity profile at the outlet is unknown. Hence, a stable energy balance cannot be guaranteed a priori during backflow, i.e., when $\boldsymbol{u}\cdot\boldsymbol{n} < 0$ on $\Gamma_{\text{out}}$. This issue typically arises when cutting the physical domain and imposing Neumann boundary conditions (as in problem (1)), which do not consider the physical convective effects present in the neglected parts of the physical domain. In practice, this might cause large unphysical oscillations in the velocity near the outlet, compromising the stability, the feasibility and the reliability of the numerical simulations (see, e.g., Fig. 1).

Different treatments to overcome this problem have been already proposed in the literature. A first group of methods consist in imposing additional constraints on the velocity field at the Neumann boundary, e.g. via enforcing the shape of the velocity profile. Due to the global mass conservation, this directly controls the magnitude of the velocity field on $\Gamma_{\text{out}}$, ensuring the overall stability. However, this shape constraint is usually imposed through Lagrange multipliers [1], which might involve considerable modifications of the numerical solver and might considerably increase the overall computational cost. Simpler variants consist in constraining only the direction of the flow on the Neumann boundaries, for example enforcing the outlet velocity to be normal to the boundary. This approach can reduce the oscillations, but it does not necessarily eliminate them [2].

A second group of methods is based on achieving stability imposing the total outlet pressure $\boldsymbol{\sigma}(\boldsymbol{u}, p)\boldsymbol{n} = p_{\text{out}} + \rho|\boldsymbol{u}|^2/2$ at the open boundary, hence modifying the Neumann boundary condition $(1)_5$ (see, e.g., [3]). However, this might lead to unphysical solutions [4]. Inspired from [5,6], a similar strategy consists in modifying the Neumann boundary condition $(1)_5$ as

$$\boldsymbol{\sigma}(\boldsymbol{u}, p)\boldsymbol{n} = -\bar{p}_{\text{out}}\boldsymbol{n} + \beta\frac{\rho}{2}|\boldsymbol{u}\cdot\boldsymbol{n}|_-\boldsymbol{u} \quad \text{on } \Gamma_{\text{out}}, \tag{3}$$

with

$$|\boldsymbol{u}|_- := \frac{\boldsymbol{u} - |\boldsymbol{u}|}{2}, \quad \beta \geqslant 0, \tag{4}$$

so that the Neumann condition is only perturbed in the presence of backflow. In particular, two variants of this method were recently reported. The first, developed in [2,7] in the context of hemodynamics, is based on the choices $\beta \leqslant 1.0$ and $\bar{p}_{\text{out}} = p_{\text{out}}$, and will be denoted in what follows as *inertial stabilization*. The second, using $\beta = 1$ and $\bar{p}_{\text{out}} = p_{\text{out}} + f(U, Q)\rho/2$, has been proposed in [8] for respiratory mechanics and it is also suitable for blood flows. Here, $f(U(\mathbf{x}), Q)$ corresponds to an approximation of $|\boldsymbol{u}\cdot\boldsymbol{n}|_-\boldsymbol{u}$, based on a assumed velocity profile $U(\mathbf{x})$ on the open boundary and a given – or computed – flux $Q$, also allowing the simultaneous imposition of pressure and flows rates. Note that, for all these techniques, the global stability is ensured if $\beta = 1$, according to Eq. (2).

The aim of this work is to propose a new stabilized formulation, based on a local regularization of the fluid velocity along the tangential directions on the Neumann boundaries. The stabilization consists in a symmetric penalization of the tangential variation of the outlet velocity, proportional to the amount of backflow, and it does not require any assumption on the velocity profile.

The rest of the paper is organized as follows. The tangential regularization is introduced in Section 2. In Section 3 the performance of the method is assessed through extensive numerical examples in blood flow regimes. In order to explain the behavior observed in the numerical examples, Section 4 discusses a possible analytical estimation of the stabilization parameter in terms of the mesh size. Finally, Section 5 draws the conclusions.