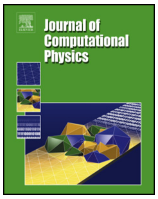
Journal of Computational Physics 261 (2014) 83–105

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp



A robust and accurate outflow boundary condition for incompressible flow simulations on severely-truncated unbounded domains

S. Dong [a,*], G.E. Karniadakis [b], C. Chryssostomidis [c]

[a] *Department of Mathematics, Purdue University, United States*
[b] *Division of Applied Mathematics, Brown University, United States*
[c] *Department of Mechanical Engineering, Massachusetts Institute of Technology, United States*

ARTICLE INFO

Article history:
Received 29 June 2013
Received in revised form 17 December 2013
Accepted 19 December 2013
Available online 3 January 2014

Keywords:
Outflow
Boundary condition
Unbounded domain
High Reynolds number
Spectral element

ABSTRACT

We present a robust and accurate outflow boundary condition and an associated numerical algorithm for incompressible flow simulations on unbounded physical domains, aiming at maximizing the domain truncation without adversely affecting the flow physics. The proposed outflow boundary condition allows for the influx of kinetic energy into the domain through the outflow boundaries, and prevents un-controlled growth in the energy of the domain in such situations. The numerical algorithm for the outflow boundary condition is developed on top of a rotational velocity-correction type strategy to de-couple the pressure and velocity computations, and a special construction in the algorithmic formulation prevents the numerical locking at the outflow boundaries for time-dependent problems. Extensive numerical tests for flow problems with bounded and semi-bounded physical domains demonstrate that this outflow boundary condition and the numerical algorithm produce stable and accurate simulations on even severely truncated computational domains, where strong vortices may be present at or exit the outflow boundaries. The method developed herein has the potential to significantly expedite simulations of incompressible flows involving outflow or open boundaries, and to enable such simulations at Reynolds numbers significantly higher than the state of the art.

© 2013 Elsevier Inc. All rights reserved.

1. Introduction

Outflows, or more generally open portions of the boundary, where the field variables are unknown and need to be computed, are encountered in wakes, jets, boundary layers, arterial trees, and many other flow problems. How to deal with outflows for incompressible flow simulations is a challenging and open issue. On the one hand, an ideal method would allow the flow and any information carried with it to exit the domain without adverse upstream effects [46]. On the other hand, it should allow for stable computations regardless of the location of the outflow boundary, even at high Reynolds numbers. Simple analysis for a one-dimensional advection–diffusion system with the standard Neumann outflow condition shows that, to maintain good solution accuracy, the length of the domain should scale linearly with the Reynolds or Peclet number, which is obviously a severe limitation.

---

[*] Corresponding author.
E-mail address: sdong@math.purdue.edu (S. Dong).

0021-9991/$ – see front matter © 2013 Elsevier Inc. All rights reserved.
http://dx.doi.org/10.1016/j.jcp.2013.12.042

We briefly summarize the existing work on the outflow or open flows in the literature, restricting our attention to incompressible flows. For the outflow issues with compressible flows, some recent reviews are [3,15]; see also the references therein. For more general problems one can refer to [52] for a comprehensive review.

Given the importance of this topic, it is not surprising that a large volume of literature exists on the outflow conditions for incompressible flows. The techniques for dealing with outflow boundaries have been reviewed, up to the mid-1990s, in [46]; see also [17] and the references therein. The types and the ease in implementation of the outflow boundary conditions are closely associated with the spatial discretization schemes. The traction-free boundary condition (or its variants, e.g. no-flux condition) at the outflow is one of the most commonly used [50,14,10,33,1,46,2,21,34], especially for finite element-type discretizations. The convective (or radiation) boundary condition (see e.g. [39,17,30,38,11,45]) has often been used with finite difference or finite volume type discretizations.

A number of other types of outflow conditions have been proposed based on various strategies over the years. In [26] a non-reflecting condition was developed through a factorization of the wave operator and by making an analogy between the incompressible Navier–Stokes equation and the factorized wave equation. A free boundary condition at the outflow was proposed in [40], where the unknown surface integral term at the outflow boundary in the weak formulation of the Navier–Stokes equation was moved back to the left hand side and incorporated into the coefficient matrix. This is often referred to as “do-nothing” boundary condition or “no-boundary-condition” condition. This boundary condition has been analyzed in [19,43], and it is equivalent to imposing the condition that the $(N+1)$-st derivative of the field variable, where N is the polynomial degree of the finite elements, should vanish at a point near the outflow boundary. Another outflow condition, which was proposed in [27] and aims to suppress a boundary layer at the open boundary, requires a normal derivative of a high order of some independent variable to vanish at the outflow boundary. In the method of [23], the velocity at the outflow boundary is determined by extrapolating the velocity from the interior of domain and by utilizing an asymptotic velocity field at infinity, while the pressure at the outflow boundary is determined by using the traction-free condition. In [35], a velocity–pressure formulation of the Navier–Stokes equations was used, where a pressure Poisson equation replaces the velocity divergence-free condition (see also e.g. [17,34]), and then a set of conditions for both the outflow and inflow boundaries are obtained by solving an eigen-value problem at the boundaries [36]. An augmented Lagrangian method was developed in [31], where the velocity profiles at the outflow boundaries are constrained to achieve certain desired properties. In [42,41], an equation for the pressure at the outflow boundary is derived involving derivatives along the tangential direction of the outflow boundary. It is then proposed that the pressure condition at the outflow boundary be determined by solving this equation. Defective boundary conditions, in which only the averaged quantities such as the flow rate or pressure drop are available on portions of the domain boundary, are important for many applications such as blood flow simulations in arterial networks [20]. This type of problems has been considered in [24]; see also [12].

In production simulations a common observation with outflows is that, especially at high and moderate Reynolds numbers (or even at low Reynolds numbers in certain situations), the computation would become unstable when strong vortices are present at or exit the outflow boundaries [6,7]. Owing to the lack of effective techniques, a usual remedy for this problem is to employ a large computational domain, e.g. by placing the outflow boundary far away from the domain of interest, such that the presence of outflow boundary would not significantly disturb the flow, and more importantly, the vortices generated upstream would be sufficiently dissipated before reaching the outflow boundary. For example, in the three-dimensional direct numerical simulation of turbulent flow past a circular cylinder at Reynolds number $Re = 10{,}000$ [7], a wake region of length $50D$ (D being the cylinder diameter) in the streamwise direction has been employed. Even though the far-wake region was not of interest in that study, such a large computational domain is essential for numerical stability in order for the vortices to be sufficiently dissipated before reaching the outflow boundary at that Reynolds number. This practice has several drawbacks: the large computational domain requires a large mesh and induces a large computational cost, and much of the grid resolution would be wasted in the far regions which are of little or no physical interest. In addition, this strategy is not scalable with respect to the Reynolds number [7].

In this paper, we present a robust and accurate outflow boundary condition, and the associated numerical algorithm, that allow for stable simulations of incompressible flows on even severely truncated computational domains, where strong vortices may be present at or exit the outflow boundaries. Fig. 1 shows a snapshot of the instantaneous velocity field and pressure distribution (color contours) of the flow past a square cylinder at Reynolds number $Re = 10{,}000$ from a two-dimensional (2-D) simulation using our method. The computational domain has a streamwise length of only $5.5D$ in the cylinder wake, where D is the cylinder dimension. One can clearly observe the strong vortices passing through the outflow boundary. The new outflow boundary condition is designed in such a fashion that the influx of kinetic energy into the domain through the outflow boundaries, if any, will not cause un-controlled growth in the energy of the domain. The algorithm for dealing with this outflow boundary condition is developed on top of a rotational velocity-correction type strategy [8] for de-coupling the pressure and velocity computations in the incompressible Navier–Stokes equations. In the pressure and velocity sub-steps, different types of discrete conditions are imposed for the pressure and the velocity on the outflow boundaries. A special construction in the algorithmic formulation obviates the numerical locking on the outflow boundaries for time-dependent problems.

The method developed herein can potentially enable the use of a much smaller computational domain for incompressible flow simulations at high Reynolds numbers. With the current method the domain size will only be restricted by considerations of physical accuracy, not by the numerical stability associated with the outflow boundary. Since one can pack all the

Download English Version:

https://daneshyari.com/en/article/518348

Download Persian Version:

https://daneshyari.com/article/518348

Daneshyari.com