Computers and Fluids 134–135 (2016) 177–189

Contents lists available at ScienceDirect

Computers and Fluids

journal homepage: www.elsevier.com/locate/compfluid

A viscous sponge layer formulation for robust large eddy simulation of thermal plumes

Chandra Shekhar Pant[a], Amitabh Bhattacharya[a,b,*]

[a] Department of Mechanical Engineering, I.I.T. Bombay, Powai, Mumbai, 400076, India
[b] Interdisciplinary Programme In Climate Studies, I.I.T. Bombay, Powai, Mumbai, 400076, India

ARTICLE INFO

Article history:
Received 31 August 2015
Revised 3 May 2016
Accepted 20 May 2016
Available online 24 May 2016

Keywords:
Large eddy simulation
Viscous sponge layer
Free shear flows
Pure thermal plume
Turbulence
Numerical stability

ABSTRACT

Numerical simulations of pure turbulent plumes for incompressible flow are known to be unstable, primarily due to flow structures being advected into or out of the computational domain at the inflow/outflow/convective (IOC) boundary. To address this issue, we introduce sponge layers adjacent to the IOC boundaries, in which viscous damping of the flow field is carried out only in directions tangential to the boundary. This allows us to introduce a sharp jump in artificial viscosity inside the sponge layer, without introducing any spurious source terms in the momentum equation. This large viscosity damps the small scale flow structures at the IOC boundaries, allowing smooth exit of flow structures. Several numerical tests are carried out to show that results from Large Eddy Simulation (LES) of a pure thermal plume using the above viscous sponge layer scheme are independent of the size of the domain. The results are also insensitive to the thickness of the sponge layer. Data from our LES simulations match analytical self-similar results, as well as prior LES and DNS simulations. The interaction of a single vortex ring with the sponge layer is also studied, and it is found that the sponge layer does not significantly affect parts of the vortex ring lying outside the sponge layer, while smoothening out parts of the vortex ring lying inside the sponge layer, allowing for smooth exit of the vortex ring from the computational domain.

1. Introduction

Pure thermal plumes commonly exist in both natural and industrial processes. Thermal plumes are especially important in atmospheric dynamics and geophysical flows [6,22,23], and are therefore extensively studied numerically, experimentally and analytically [1,9,15,19].

Proper application of inflow/outflow and convective (denoted hereon as IOC) boundary conditions has always been a challenging issue in numerical simulations of free shear flows, especially for pure thermal plumes. For compressible flow solvers, several robust schemes exist for applying IOC boundary conditions [2]. For incompressible flow solvers, while several schemes do exist for applying these boundary conditions, they each have their own severe shortcomings, as listed below.

Standard Neumann boundary condition is found to cause significant non-physical fluctuations in the pressure field [3,12].

The convective boundary condition, which states that flow structures should be freely convected outside the domain, was introduced by Orlanski [13]:

$$\frac{\partial \phi}{\partial t} = -c\frac{\partial \phi}{\partial n}, \tag{1}$$

where $\phi(\mathbf{x}, t)$ is a scalar field, c is the phase velocity, and n is the coordinate normal to the boundary. Several modifications to the convective boundary condition have since been proposed [10,16,18]. However, Hattori *et al.* [7] demonstrated that, for simulations involving thermal plumes, several of these schemes are unstable beyond a certain time.

In recent work by Dong et al. [5], a new IOC boundary condition was proposed for flow around bluff bodies. It was hypothesized that inflow/outflow of flow structures at the boundaries can cause energy addition into the flow domain. As a solution, the traction at the boundary due to pressure and viscous stresses were equated to a term proportional to the inflow kinetic energy. However, so far, this method has not been used to simulate thermal plumes.

Craske *et al.* [3], inferred that the convective open boundary condition is unsuitable for direct numerical simulation of incompressible turbulent jets and plume. They further pointed out that this boundary condition modifies the spreading rate, leads to results that are domain dependent, and does not prevent instabilities from propagating into the domain. They proposed an improved

* Corresponding author.
E-mail address: bhattach@gmail.com (A. Bhattacharya).

http://dx.doi.org/10.1016/j.compfluid.2016.05.019
0045-7930/

version of convective boundary condition in which proper care is taken for the phase velocity (of Eq. (1)) which ensures reflection free boundary condition in plume and jets. In addition to this, viscous terms are added to the right hand side of Eq. (1) in accordance to the Miyanchi *et al.* [12]. They successfully show a good agreement of direct numerical simulation of jet and plume with the truncated domain and the new proposed outflow boundary condition. This method however requires statistical axisymmetry of the flow, and therefore cannot be applied to more complicated flows (e.g. involving multiple thermal plumes).

Another technique, which is used for incompressible flows involving IOC boundary conditions, involves application of a sponge layer near the boundary. An example of such a technique is the work Mittal and Balachandar [11], in which viscous terms, as well as source terms in the pressure equation, are attenuated in a sponge region. This technique requires smooth variation of the attenuation function, and has again not been demonstrated for a thermal plume.

In this work, we propose a novel method for applying viscous sponge layers adjacent to boundaries where IOC conditions are being applied. In our method, we add sponge layers adjacent to the IOC boundaries, in which additional viscous terms are added to the momentum equation. The additional viscous damping of momentum is carried out only in directions parallel to the boundary. This ensures that the flow structures are smoothened out in directions tangential to the boundary. Continuity equation additionally ensures that, inside the sponge layer, the velocity normal to the boundary is smooth along the outflow direction, ensuring a temporally smooth phase velocity at the boundary. Our method thus allows us to introduce sharp jumps in viscosity inside the sponge layer, without incurring spurious source terms at the jump location itself.

The main goal of this work is to understand the effect of the proposed sponge layer on both mean flow statistics, as well as individual flow structures. This manuscript is therefore organized as follows. We first describe our methodology in Section 2. After describing the flow domain and governing equations (Section 2.1), we describe our sponge layer formulation in Section 2.2, as well as details of the numerical discretization used to solve the governing equations (Section 2.3). In the results section (Section 3), we first examine mean and instantaneous flow field of a pure thermal plume at different Richardson numbers. We start by examining results with and without the sponge layer (Sections 3.1 and 3.2). In Section 3.2, we also illustrate the effect of the sponge layer on the flow structures in the thermal plume located near the boundary. In Section 3.3, we compare our results with the sponge layer to available DNS and LES data for small domain sizes, where self-similarity assumptions do not hold for the mean flow field. In Section 3.4, we study the dependence of mean flow statistics on domain size and aspect ratio, for large flow domains, where self-similar solutions exist. Here, we also report the sensitivity of the results with respect to the thickness of the viscous sponge layer, and compare mean statistics with analytical self-similar solutions. Finally, in Section 3.5, we study the effect of sponge layer on a single vortex ring.

2. Methodology

2.1. Flow domain and governing equations

Our computational domain consists of a box with dimensions $L_x^* \times L_y^* \times L_z^*$. We solve continuity, momentum and energy equations for an incompressible fluid inside the computational domain $\mathcal{D}$ (Fig. 1), using Boussinesq approximation to model buoyancy force. The dimensional velocity field $\mathbf{u}^*(\mathbf{x}^*, t^*)$, position $\mathbf{x}^*$, temperature field $T^*(\mathbf{x}^*, t^*)$, time t^*, and pressure field $P^*(\mathbf{x}^*, t^*)$ are

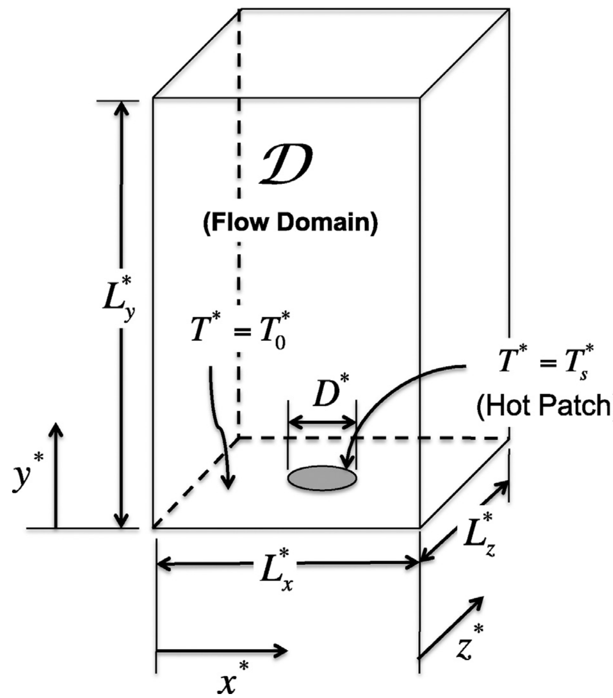

Fig. 1. Schematic of computational box,

non-dimensionalized using g^*, D^* and $\Delta T^* = T_s^* - T_0^*$. Here g^* is magnitude of acceleration due to gravity, D^* is the diameter of the hot patch that generates the plume, T_s^* is the temperature of the hot patch, and T_0^* is the temperature of the surface surrounding the hot patch. The non-dimensional variables are then $\mathbf{u} = \mathbf{u}^*/\sqrt{g^* D^*}$, $T = (T^* - T_0^*)/(T_s^* - T_0^*)$, $\mathbf{x} = \mathbf{x}^*/D^*$, $t = t^*\sqrt{g^*/D^*}$, $L_x = L_x^*/D$, $L_y = L_y^*/D$ and $L_z = L_z^*/D$. The non-dimensionalized equations are given by:

$$\nabla \cdot \mathbf{u} = 0 \tag{2}$$

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla P + (\mathrm{Ri}_0) T \mathbf{j} - \nabla \cdot \sigma^t + \mathbf{b}^s \tag{3}$$

$$\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T = -\nabla \cdot \mathbf{Q} \tag{4}$$

Here, $\mathrm{Ri}_0 = \beta \Delta T^*$ is the Richardson number, which couples temperature and velocity evolution equations, $\mathbf{j}$ is the unit vector in the positive y direction, σ^t is the stress tensor due to unresolved turbulent scales, $\mathbf{b}^s$ is the body force due to artificial viscosity in the sponge layer, and $\mathbf{Q}$ is the heat flux vector. It should be noted that Ri_0 is the Richardson number at the inlet, which is different from it's value in the flow [21]. We use a Large Eddy Simulation (LES) model [17] to close the subgrid stress and energy flux in terms of $\mathbf{u}$ and T. For free shear flows, LES models assume that the flow has infinite Reynolds number, and that the filter width (or size of the grid cell) lies inside the inertial range of the turbulent energy spectrum. Therefore, for a fixed flow geometry, the Richardson number (Ri_0) is the only free non-dimensional parameter present in the above equations. In our LES simulations, we use the classical assumption that the large (resolved) scales of velocity, $\mathbf{u}$, and temperature field, T, contain most of the energy, while the small (unresolved) scales mostly dissipate this energy. The effect of the small scales on the large scales is modeled via the subgrid stress tensor and the heat flux vector using the standard Smagorinsky–Lilly model [17]:

$$\sigma_{ij}^t = \frac{\sigma_{kk}^t}{3}\delta_{ij} - 2\nu_t S_{ij} \tag{5}$$