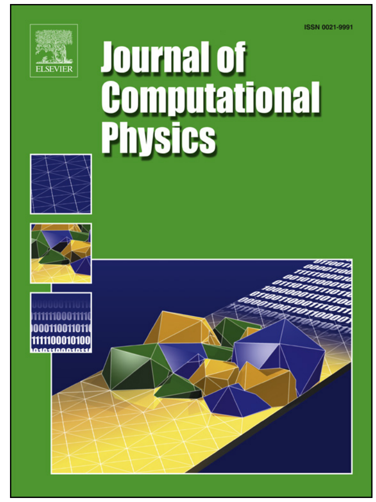
Accepted Manuscript



Multiple-resolution scheme in finite-volume code for active or passive scalar turbulence

Kai Leong Chong, Guangyu Ding, Ke-Qing Xia

PII: S0021-9991(18)30613-2
DOI: https://doi.org/10.1016/j.jcp.2018.09.019
Reference: YJCPH 8266

To appear in: *Journal of Computational Physics*

Received date: 7 May 2018
Revised date: 5 September 2018
Accepted date: 8 September 2018

Journal of Computational Physics (2018)

Contents lists available at ScienceDirect

Journal of Computational Physics

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

Multiple-resolution scheme in finite-volume code for active or passive scalar turbulence

Kai Leong Chong[a,1,*], Guangyu Ding[a], Ke-Qing Xia[a,**]

[a]*Department of Physics, The Chinese University of Hong Kong, Shatin, Hong Kong, China*

ARTICLE INFO

Article history:
Received x May 2018
Received in final form x x 2018
Accepted x x 2018
Available online x x 2018

Communicated by xx

ABSTRACT

In scalar turbulence it is sometimes the case that the scalar diffusivity is smaller than the viscous diffusivity. The thermally-driven turbulent convection in water is a typical example. In such a case the smallest scale in the problem is the Batchelor scale l_b, rather than the Kolmogorov scale l_k, as $l_b = l_k/Sc^{1/2}$, where Sc is the Schmidt number (or Prandtl number in the case of temperature). In the numerical studies of such scalar turbulence, the conventional approach is to use a single grid for both the velocity and scalar fields. Such single-resolution scheme often over-resolves the velocity field because the resolution requirement for scalar is higher than that for the velocity field, since $l_b < l_k$ for $Sc > 1$. In this paper we put forward an algorithm that implements the so-called multiple-resolution method with a finite-volume code. In this scheme, the velocity and pressure fields are solved in a coarse grid, while the scalar field is solved in a dense grid. The central idea is to implement the interpolation scheme on the framework of finite-volume to reconstruct the divergence-free velocity from the coarse to the dense grid. We demonstrate our method using a canonical model system of fluid turbulence, the Rayleigh-Bénard convection. We show that, with the tailored mesh design, considerable speed-up for simulating scalar turbulence can be achieved, especially for large Schmidt (Prandtl) numbers. In the same time, sufficient accuracy of the scalar and velocity fields can be achived by this multiple-resolution scheme. Although our algorithm is demonstrated with a case of an active scalar, it can be readily applied to passive scalar turbulent flows.

*Corresponding author: Email.: stevenphycuhk@gmail.com
**Corresponding author: Email.: kxia@cuhk.edu.hk
[1]Present address: Physics of Fluids Group and Max Planck Center Twente, MESA+Institute, J. M. Burgers Centre for Fluid Dynamics, University of Twente, 7500 AE Enschede, Netherlands