Advances in Applied Mathematics and Mechanics
Adv. Appl. Math. Mech., Vol. **15**, No. 3, pp. 769-785

DOI: 10.4208/aamm.OA-2022-0147
June 2023

Parallel Domain Decomposition-Based Solver for the Simulation of Flow over an Ahmed Reference Model

Zhengzheng Yan[1], Rongliang Chen[1], Chao Wang[2], Lei Xu[1,*] and Jingzhi Li[3,*]

[1] *Shenzhen Institutes of Advanced Technology, Chinese Academy of Sciences, Shenzhen 518055, China*
[2] *Faculty of Computational Mathematics and Cybernetics, Shenzhen MSU-BIT University, Shenzhen 518172, China*
[3] *Department of Mathematics & National Center for Applied Mathematics Shenzhen & SUSTech International Center for Mathematics, Southern University of Science and Technology, Shenzhen 518055, China*

Received 29 May 2022; Accepted (in revised version) 12 August 2022

Abstract. The Ahmed model is a standard bluff body used to study the flow behavior around an automobile. An important issue when investigating turbulent flow fields is the large computational load driven by accurate prediction approaches, such as the large eddy simulation model. In this paper, we present a powerful domain decomposition method-based parallel solver to efficiently utilize existing supercomputer resources. The 3D unsteady incompressible Navier–Stokes equations with a subgrid-scale (SGS) fluid model are discretized on a pure unstructured tetrahedral grid by a stable P_1-P_1 finite element method in space, while an implicit second-order backward differentiation formula is employed for the time discretization. We then solve the nonlinear algebraic system by means of the Newton–Krylov–Schwarz method by imposing a restricted additive Schwarz (RAS) right preconditioner for the parallel setting. We validate the proposed method toward the comparison of the flow field, including the velocity profiles and flow structures, with experimental investigations, and we show the parallel efficiency and scalability of the solver with up to 8192 processors.

AMS subject classifications: 76D05, 76F65, 65M55, 65Y05

Key words: Aerodynamics, Ahmed model, full Navier-Stokes equations, Newton-Krylov-Schwarz algorithm, parallel computing.

*Corresponding author.
Emails: zz.yan@siat.ac.cn (Z. Yan), rl.chen@siat.ac.cn (R. Chen), wangchao@smbu.edu.cn (C. Wang), lei.xu@siat.ac.cn (L. Xu), li.jz@sustech.edu.cn (J. Li)

1 Introduction

The aerodynamic performance of ground vehicles is directly reflected in the external shapes. To investigate the flow behavior around a car with a particular type of rear shape, Ahmed [1] proposed a reference model for the first time to conduct detailed research on a typical flow structure. After decades of experimental investigations on the original [2] and improved model, for instance, different rear slant angles [3] and attracting passive [4–6] or active control sets [7,8], a large and complete wind tunnel experiment data set has been formed, which is frequently employed as a benchmark in vehicle aerodynamics [9].

CFD-based numerical investigations of the flow around the Ahmed body have also been developed in the past two decades. The flow around the Ahmed body is unsteady and fully turbulent at the wake of the model where recirculation and separation occur, and direct numerical simulation (DNS) is rarely implemented due to the high Reynolds number and is not practicable for now [10]. The challenge of the corresponding numerical investigations is to select quantified turbulence at multiple scales to arrive at a good balance between physical accuracy and computational time. Three main categories of numerical approaches, Reynolds averaged Navier–Stokes (RANS) for mean flow quantities [11], more accurate detached eddy simulation (DES) [12] and large eddy simulation (LES) [13,14], have been performed to evaluate the wake behind an Ahmed body.

In most circumstances, LES can provide more accurate solutions of turbulent flows than other approaches. One of the most important features of the LES model is that the computational mesh should be sufficiently refined to guarantee the filter size. However, a higher mesh resolution incurs a high requirement for hardware resources. With current computational availability, i.e., the latest supercomputers, this increase in resource demands is often justifiable. How to efficiently utilize existing powerful hardware resources to meet the demand of high accuracy and timeliness has becomed a focus of research in the automotive industry. Notably, several studies we have reviewed were all carried out on hundreds of processors. Evstafyeva et al. [15] used 384 cores on a parallel cluster to analyze the wake flow behind a square back Ahmed body. Buscariolo [16] performed a validation study using high-order methods in the open source code Nektar++ with 432 CPUs. Apart from these, most of the previous literatures were focused on numerical models and employs commercial solvers, such as ANSYS Fluent [17–19] and STAR-CCM+ [20], without reporting the detailed computing resources used or the parallel performance of the algorithm. However, a solver with high parallel performance is mandatory because of the computational load issue in the more computationally demanding LES-related simulations. To our knowledge, no related simulations have yet achieved more than one thousand cores.

The main objective of the current work is to present a powerful domain decomposition method-based parallel solver to accommodate the vast computation load driven by a suitable and reliable LES of flow over an Ahmed body. For this purpose, a subgrid-scale (SGS) model for practical LES is first proposed to capture the turbulent flow features at