Computers and Fluids 123 (2015) 122–145

Contents lists available at ScienceDirect

Computers and Fluids

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

Validation of OpenFOAM numerical methods and turbulence models for incompressible bluff body flows

E. Robertson[a], V. Choudhury[b], S. Bhushan[a,b,*], D.K. Walters[a]

[a] Department of Mechanical Engineering, Mississippi State University, Starkville, MS 39759, United States
[b] Center for Advanced Vehicular Systems, Mississippi State University, Starkville, MS 39759, United States

ARTICLE INFO

Article history:
Received 12 November 2014
Revised 27 July 2015
Accepted 23 September 2015
Available online 3 October 2015

Keywords:
OpenFOAM
Verification and validation
Numerical methods
Turbulence models

ABSTRACT

A verification and validation study was performed using the open source computational fluid dynamics solver OpenFOAM version 2.0.0 for incompressible bluff body fluid flows. This includes flow over a backward facing step, a sphere in the subcritical regime, and delta wing with sharp leading edge. The study investigates solver scalability, and accuracy of numerical methods and turbulence models available in the solver. Grid verification study shows mostly monotonic convergence with averaged grid uncertainty <5% for integral quantities and up to 10% for local variables. The solver shows good strong scalability up to 192 processors on a mesh with 11M cells. The study identifies that the 2nd order linear upwind scheme is most efficient and accurate for Reynolds Averaged Navier Stokes (RANS) simulations, while the 1st/2nd order blended limited linear scheme is best for simulations employing hybrid RANS/Large Eddy Simulation (HRL). PIMPLE and SIMPLE pressure–velocity coupling methods are identified to be best for HRL and RANS simulations, respectively. The validation study showed that drag and mean velocity predictions compared within 5% of the experimental data, whereas larger errors were predicted for turbulent kinetic energy and instability frequency predictions. OpenFOAM predictions compared within 6% of FLUENT results for backward facing step and sphere cases, and performed better than the latter for the delta wing vortex breakdown predictions. Overall, OpenFOAM is found to be a reliable research solver; however, it is more sensitivity to grid quality than FLUENT, which needs to be further investigated.

1. Introduction

The OpenFOAM computational toolbox [1–3] is a free, open-source software package capable of simulating a wide variety of fluid flow processes. The default toolbox includes over 80 solver modules, each tailored to flows ranging from simple incompressible and compressible flows to chemical reactions and fluid–structure interactions. In addition, over 170 utilities are available for grid generation and pre- and post-processing.

The popularity of OpenFOAM for various applications is rapidly growing. In recent years, OpenFOAM users worldwide have published several notable studies in the fields of computational fluid dynamics (CFD) [4–14], computational heat transfer, fluid–structure interaction, and multiphase flow. The studies have focused on a wide range of application areas, such as atmospheric boundary layer modeling for wind turbine studies, propellant combustion modeling and diesel spray ignition, turbomachinery and industrial diffusers, cavitation in flow around a submarine hull, and 2-D flow around a bridge deck (to name a few). Readers are referred to Choudhury [15] for a detailed review and references. The following provides a summary of the OpenFOAM validations and applications for CFD studies.

de Villiers [4] implemented and tested wall treatment approaches for detached eddy simulation (DES) and large eddy simulation (LES) models available in OpenFOAM. The model implementation was validated for a series of cases, including channel flow, flow through an asymmetric diffuser and flow over a blunt body, and was applied to study aero-acoustic noise levels in flow over automobiles. The study reported very good agreement between experimental data and computational results. Doolan [5] studied the flow and noise predictions for 2D incompressible flow past tandem cylinders. The study was performed using k–ε unsteady RANS (URANS) model, and the validation focused mostly on the mean and unsteady flow and noise level predictions against experimental data. The study concluded that OpenFOAM can provide accurate noise source data for low Mach number bluff body aeroacoustic flows.

Muntaen and Nilsson [6] performed numerical investigations of unsteady swirling flow in a conical diffuser using the standard k–ε model. The OpenFOAM predictions were compared with ANSYS/fluent (FLUENT henceforth) results, and validated against experimental data. The study aimed at studying the effects of the vortex rope

* Corresponding author. Tel.: +1 662 325 9612; fax: +1 662 325 5433.
E-mail address: shanti@cavs.msstate.edu (S. Bhushan).

http://dx.doi.org/10.1016/j.compfluid.2015.09.010
0045-7930/

and explicating the physics of the helical breakdown phenomenon. They concluded that OpenFOAM accurately predicts the fundamental frequency and higher harmonics of the vortex rope in the throat and in the middle of the diffuser.

Nilsson [7] performed simulations using the k–ε model to predict flow in a Kaplan water turbine runner and draft tube, and for the unsteady swirling flow in a combustor. The study reported results generally agreeable with experimental observations. They reported that the results from the draft tube case were almost identical to those of the commercial CFX-5 solver, and those for the combustor case were comparable to FLUENT results.

A variety of studies and symposiums associated with renewable energy (i.e. aerodynamics of wind turbines) have also been conducted using OpenFOAM. Notably, Panjwani et al. [8] performed wind turbine simulations to predict the power deficit expected in the wind turbine wake. They reported that OpenFOAM simulations perform quite well for the prediction of wake deficit, and wind farm power generation. Churchfield et al. [9] performed LES calculations for flow in the atmospheric boundary layer with shear driven to moderately convective stability conditions. The study focused on the analysis of second-order turbulence statistics in the boundary layer, and included comparison with benchmark LES data available in the literature. The study identified some drawbacks in the available LES models available in the solver.

Verhoeven [10] performed improved delayed DES (IDDES) simulations of flow over a NACA 0012 airfoil at varying angles of attack and for an airfoil with blunt trailing edge to study trailing edge noise signatures. The study emphasized that the interaction of turbulent structures with the trailing edge is the primary cause of noise generation in most engineering applications. The study reported encouraging initial results, and reveled that trailing edge is the primary cause of noise generation in most engineering applications. The study also reported that the available IDDES model should be more extensively tested with simpler cases to ascertain accuracy. Flores et al. [11] successfully applied OpenFOAM for buoyant atmospheric flow simulations. The simulations were performed using DES to study contaminant transport inside large open pit mines using both idealized and real topographies. The study provided reasonable predictions, and identified the key role played by buoyant currents in dispersion of contaminants inside and around such mines.

The above studies demonstrate that many users have applied the OpenFOAM toolbox to industrial problems, including some implementation and validation of their own models. However, very little research has been performed with the specific objective of testing and validating OpenFOAM itself, without any code modification or re-implementation, for fundamental problems. Notably, Lysenko et al. [12] studied turbulent separated flows over planar bluff bodies, such as circular and triangular cylinders in a channel using compressible k–ε model. The study shows OpenFOAM has good parallel performance up to 1024 cores and provides results in agreement with experimental data and other numerical solutions. Lysenko et al. [13] also studied a few of the LES models available in OpenFOAM for flow over a circular cylinder. They reported good agreement with experimental data and concluded that OpenFOAM numerical methods are sufficiently accurate for LES calculations.

The present authors have performed several validation studies as a precursor to this paper focusing on canonical and complex incompressible flows using the OpenFOAM version 2.0.0 framework released in June 2011. The earliest study, by Robertson et al. [14], involved validation of the solver focusing on 2-D lid-driven cavity flow, 2-D pipe flow at a T-junction, and surface pressure and skin friction validation of flow over a DARPA SUBOFF geometry. The study reported good agreement with similar numerical studies and experimental data. Choudhury [15] performed a thorough validation of available numerical methods and turbulence models using a backward facing step geometry. Choudhury [15] also performed a preliminary study of flow over a delta wing at different angle of attack. Robertson et al. [16] extended the above study, to elucidate the vortex breakdown phenomena over a delta wing using delayed DES (DDES) and k–ω SST models.

This study combines and expands upon the previous efforts, and serves as a user guideline for effective numerical method and model combinations for OpenFOAM applications. It also serves as an exposition of several issues facing the OpenFOAM community with regard to the often mystifying numerical implementations available in the framework. This study accomplishes this by reporting on solver scalability, and verification and validation of OpenFOAM numerical methods and turbulence models for incompressible bluff body flows. The test cases used in the study include flow over a backward facing step (BFS), sphere and sharp leading edge delta wing. The results are compared to FLUENT predictions, and with available experimental and/or numerical data. The following section provides a brief description of the numerical methods and models available in OpenFOAM. Section 3 provides details of the test cases used in the study. The results are discussed in Section 4, and conclusions are drawn in Section 5.

2. OpenFOAM numerical method and models

OpenFOAM uses the finite volume method for numerical representation of the equations governing fluid motion and the message passing interface (MPI) method for parallel computing. The toolbox features a range of numerical schemes, methods and turbulence models. The available turbulence models range from Reynolds averaged Navier–Stokes (RANS) to hybrid RANS/LES (HRL) to LES. It is also possible to resolve all scales using direct numerical simulation (DNS). The governing equations for incompressible fluid flow are the Navier–Stokes equations:

$$\nabla \cdot \vec{\boldsymbol{U}} = 0 \tag{2.1}$$

$$\frac{\partial \vec{\boldsymbol{U}}}{\partial t} + (\vec{\boldsymbol{U}} \cdot \nabla)\vec{\boldsymbol{U}} = -\nabla p + \nabla \cdot \{(\nu + \nu_T)(\nabla \vec{\boldsymbol{U}} + \nabla \vec{\boldsymbol{U}}^T)\} \tag{2.2}$$

where $\vec{\boldsymbol{U}}$ is the fluid velocity vector, p is the density-normalized pressure, ν is the kinematic viscosity of the fluid, and ν_T is the turbulent viscosity of the fluid. OpenFOAM provides three different pressure–velocity coupling methods for solving these equations: PISO (pressure implicit with split operator) [17]; SIMPLE (semi-implicit method for pressure linked equations) [18]; and PIMPLE, which is a hybrid of PISO and SIMPLE. The SIMPLEC (SIMPLE consistent) algorithm is also available, but only as part of the pressure based compressible solver rhoSimplecFoam. Since much detailed information is available in [2], the following provides only a summary of the salient aspects of the incompressible numerical methods and models.

2.1. Incompressible PISO/PIMPLE/SIMPLE solvers

The governing equations are generally solved using standard pressure–velocity coupling methodology- (1) momentum predictor, (2) pressure solver, (3) momentum corrector. The PIMPLE algorithm is a unique variation of the PISO method, where an outer correction loops, i.e., cycling over a given time step for a number of iterations, and equation under-relaxation between outer correctors are allowed for stability, as shown in Fig. 1. If no outer corrector loops are used, the algorithm is directly equivalent to the PISO method. PIMPLE solver also includes dynamic time-stepping (automatic time step adjustment to maintain a certain CFL number). The *simpleFoam* solver is based on the SIMPLE algorithm. It pursues a steady-state solution with the aid of under-relaxation factors between iterations. Equation under-relaxation helps promote diagonal dominance by boosting the influence of the owner cell terms [2].