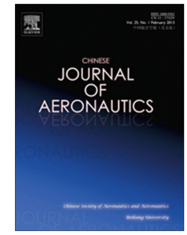
Chinese Journal of Aeronautics, (2013),26(6): 1370–1379

Chinese Society of Aeronautics and Astronautics
& Beihang University

Chinese Journal of Aeronautics

cja@buaa.edu.cn
www.sciencedirect.com



Analysis on capabilities of density-based solvers within OpenFOAM to distinguish aerothermal variables in diffusion boundary layer

Shen Chun [a], Sun Fengxian [b], Xia Xinlin [a,*]

[a] *School of Energy Science and Engineering, Harbin Institute of Technology, Harbin 150001, China*
[b] *School of Power and Energy Science, Harbin Engineering University, Harbin 150001, China*

Received 12 October 2012; revised 18 January 2013; accepted 18 March 2013
Available online 5 November 2013

KEYWORDS

Aerothermal variables;
Boundary layer;
Computational fluid dynamics (CFD);
Heat flux;
Open source;
Supersonic

Abstract Open source field operation and manipulation (OpenFOAM) is one of the most prevalent open source computational fluid dynamics (CFD) software. It is very convenient for researchers to develop their own codes based on the class library toolbox within OpenFOAM. In recent years, several density-based solvers within OpenFOAM for supersonic/hypersonic compressible flow are coming up. Although the capabilities of these solvers to capture shock wave have already been verified by some researchers, these solvers still need to be validated comprehensively as commercial CFD software. In boundary layer where diffusion is the dominant transportation manner, the convective discrete schemes' capability to capture aerothermal variables, such as temperature and heat flux, is different from each other due to their own numerical dissipative characteristics and from viewpoint of this capability, these compressible solvers within OpenFOAM can be validated further. In this paper, firstly, the organizational architecture of density-based solvers within OpenFOAM is analyzed. Then, from the viewpoint of the capability to capture aerothermal variables, the numerical results of several typical geometrical fields predicted by these solvers are compared with both the outcome obtained from the commercial software Fastran and the experimental data. During the computing process, the Roe, AUSM^{+}(Advection Upstream Splitting Method), and HLLC(Harten-Lax-van Leer-Contact) convective discrete schemes of which the spatial accuracy is 1st and 2nd order are utilized, respectively. The compared results show that the aerothermal variables are in agreement with results generated by Fastran and the experimental data even if the 1st order spatial precision is implemented. Overall, the accuracy of these density-based solvers can

* Corresponding author. Tel.: +86 451 86412148.
E-mail addresses: shenchun390@sina.com (C. Shen), xiaxl@hit.edu.cn (X. Xia).

Peer review under responsibility of Editorial Committee of CJA.

Production and hosting by Elsevier

http://dx.doi.org/10.1016/j.cja.2013.10.003

meet the requirement of engineering and scientific problems to capture aerothermal variables in diffusion boundary layer.

1. Introduction

OpenFOAM is an open source computational fluid dynamics (CFD) class library based on C++[1]. It is derived at Imperial College, London. Due to its excellent underlying data structure, programmers can pay more attention to establishing superior code structure according to a physical model. So far, except laminar flow solvers, a lot of modules with respect to other advanced physical models are assembled within OpenFOAM, such as many kinds of Reynolds average Navier–Stokes (RANS) turbulence models for incompressible and compressible flow, large-eddy simulation (LES) models for incompressible and compressible flow, combustion models, radiation models, and so on. For molecular models, the direct simulation Monte Carlo (DSMC) and the molecular dynamics methods are included. These models support parallel computation very well which can accelerate calculating process much more. With corresponding calling methods, developers can directly use the source codes corresponding to these models to satisfy their special requirements which can't be completed with commercial software. It is very convenient for developers to develop new codes and directly add many other existing models in their new codes.

Within OpenFOAM, the methods to solve Navier–Stokes equations are primarily based on the velocity–pressure coupled methods. Undergoing about 30 years of developing, these kinds of solvers have better performances on subsonic/transonic compressible flow. Recently, for supersonic and hypersonic flow, the solvers based on the density-based method for solving supersonic/hypersonic compressible flow are coming up within OpenFOAM, such as the "rhoCentralFoam" solver which applies the Kurganov and Tadmor scheme[2–4], the "DensityBasedTurbo" solvers which use the Godunov type schemes[5], and so on. However, due to lack of sufficient valid numerical examples, their efficiency and precision should be verified further in many aspects.

Many kinds of discrete schemes can capture shock wave precisely, but because of their own discrete characteristics, the numerical dissipative effects are different from each other considerably. The excessive dissipation diffuses aerothermal variables in the boundary layer severely, so the capabilities of these schemes and the corresponding programs to distinguish the boundary aerothermal variables, such as temperature and heat flux, are very different from each other.[6–10] Until now some works have already been done to testify the accuracy of the density-based solvers within OpenFOAM to capture shock wave, but their capabilities to differentiate the aerothermal variables in the boundary layer where diffusion is the overwhelming transportation manner need to be verified further. Therefore, from the point of view of boundary aerothermal variables, to testify the capabilities of the density-based solvers within OpenFOAM is the primary purpose of this study. In order to complete this task, in the present paper, firstly, the basic numerical methods and organizational architecture of the density-based solvers within OpenFOAM are analyzed. Secondly, compared with the experimental data and the computational results obtained from the noted commercial software Fastran which is widely adopted in aeronautical community, the precision of these solvers within OpenFOAM to distinguish boundary aerothermal variables is presented.

2. Numerical methods

Within OpenFOAM, DensityBasedTurbo and rhoCentralFoam are both density-based solvers which can solve supersonic and hypersonic flow problems, but the DensityBasedTurbo code assembles the famous Godunov type schemes such as Roe[11,12], AUSM(Advection Upstream Splitting Method) family (AUSM, AUSM$^+$, AUSM$^+$up[13–15]), HLLC(Harten-Lax-van Leer-Contact)[16], etc., and the time discrete schemes including the dual time scheme and the physical time step Runge–Kutta scheme. These schemes are more prevalent and universal, so in this paper the DensityBasedTurbo solver is used to analyze.

In this section, the theory of governing equations and basic numerical methods will be introduced.

2.1. Governing equation

The continuity, the momentum (Navier–Stokes), and the energy equations in vector form are:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \boldsymbol{U}) = 0 \tag{1}$$

$$\frac{\partial (\rho \boldsymbol{U})}{\partial t} + \nabla \cdot (\rho \boldsymbol{U}\boldsymbol{U}) + \nabla \cdot p - \nabla \cdot \boldsymbol{\tau} = 0 \tag{2}$$

$$\frac{\partial (\rho E)}{\partial t} + \nabla \cdot (\rho \boldsymbol{U} E) + \nabla \cdot (\boldsymbol{U} p) - \nabla \cdot (\boldsymbol{\tau} \cdot \boldsymbol{U}) + \nabla \cdot (-\lambda \nabla T) = 0 \tag{3}$$

where ρ, $\boldsymbol{U}$, T, p and E are density, velocity vector, temperature, pressure and total energy, respectively; t is time and $\boldsymbol{\tau}$ is stress tensor; λ is thermal conductivity.

2.2. Numerical schemes

Flux difference splitting (FDS) schemes, such as the Roe scheme, have very high resolution for both contact discontinuity and boundary layer. Flux vector splitting (FVS) schemes have much better robustness in capturing strong discontinuities, but have a large numerical dissipation on contact discontinuities and in boundary layer. AUSM family schemes have advantages of both FDS and FVS schemes, such as high resolution for contact discontinuity, low numerical dissipation, and high computational efficiency. Roe and AUSM, these two kinds of schemes also have high resolution for the heat flux which is mainly caused by viscosity diffusion even if the mesh density is not very high. Except these two schemes, HLLC which is quite robust and efficient but somewhat more diffusive, is also assembled internally within OpenFOAM. At present, the programs for high-speed compressible flow solvers generally adopt these three schemes, especially for unstructured grids. In this paper, the results which are obtained from