International Journal of Heat and Mass Transfer 97 (2016) 880–890

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Investigation on the mixing mechanism of single-jet film cooling with various blowing ratios based on hybrid thermal lattice Boltzmann method

Yanqin Shangguan, Xian Wang *, Yueming Li

State Key Laboratory for Strength and Vibration of Mechanical Structures, School of Aerospace, Xi'an Jiaotong University, 28 Xianning West Road, Xi'an, Shaanxi 710049, China
Shaanxi Key Laboratory of Environment and Control for Flight Vehicle, School of Aerospace, Xi'an Jiaotong University, 28 xianning west road, Xi'an, Shaanxi 710049, China

ARTICLE INFO

Article history:
Received 18 June 2015
Received in revised form 26 February 2016
Accepted 26 February 2016
Available online 15 March 2016

Keywords:
Lattice Boltzmann method
Large eddy simulation
Film cooling
Mixing mechanism
Multiple graphic processing units

ABSTRACT

Massive parallel simulation applied multiple graphic processing units (multi-GPUs) is carried out to perform an in-depth investigation on the mixing mechanism between hot crossflow and coolant jet flow in film cooling with large eddy simulation (LES) based on hybrid thermal lattice Boltzmann method (HTLBM). A coolant jet is injected at an inclined angle of $\alpha = 30^\circ$ into a turbulent flat plate boundary layer profile with a Re = 4000 free-stream Reynolds number. Three blowing ratios ranging from 0.2 to 0.8 are studied. A three-part definition on jet-crossflow-interaction region is proposed. They are shear domain, rotating domain, and dissipation domain, respectively. In shear domain, the turbulent-kinetic-energy (TKE) value is quite small and the coolant film is stable. In rotating domain, crossflow mixes with jet flow violently and coolant film loses stability gradually. The great turbulent-dissipation effect in dissipation domain causes large energy loss and disappearing of counterrotating vortex pair, which results in the poor thermal protection and coolant film collapses. Moreover, under different blowing ratios, quite different states of microscopic flow structures are presented, which causes different macroscopic heat transfer behaviors. On the other hand, the present simulation with 165 million grids is fulfilled on 9 K20M GPUs applying CUDA-MPI and a high computational performance of 896.35 MLUPS is achieved.

© 2016 Elsevier Ltd. All rights reserved.

1. Introduction

Film cooling technique is a common practice in modern gas turbine designs [1] since the gas temperature at the turbine inlet greatly exceeds the heat-resistance temperature of blade material. As for film cooling, coolant jet is injected at an angle into the hot crossflow of turbine section through jet holes drilled in the surface of blades. The coolant forms a protective layer over the turbine blade surface to protect the surface from direct exposure to hot crossflow. In order to effectively design aerodynamic and structure parameters in film cooling, the detailed understanding of the mixing mechanism between hot crossflow and coolant jet is critical.

The interaction between hot crossflow and coolant jet is extremely complex. Getting very detailed data on mixing process via experiment is still very hard, although numerous experiments [2,3] on film cooling were carried out with various precise instruments. Therefore, the numerical study with high resolution is indispensible. Over the last three decades, a wide range of numerical studies on film cooling have been conducted to investigate the film-cooling mixing mechanism and improve the cooling efficiency. Lakehal et al. [4] calculated temperature and velocity fields with various blowing ratios and the computations showed that the secondary-flow and heat-transfer mechanisms occurring in the viscosity-affected near-wall layer were difficult to be predicted precisely by the k-ε based two-layer turbulence model. Three-dimensional calculations of the flow field around a turbine blade with film cooling injection near the leading edge were performed by Theodoridis et al. [5]. It was found that the lateral jet spreading on the pressure side was under-predicted by the standard k-ε turbulence model with wall functions. By comparing with high-resolution-measurement results, Galeazzo et al. [6] made the validation of simulations ranging from simple steady-state Reynolds-averaged Navier–Stokes (RANS) to sophisticated large eddy simulation (LES), and the poor performance based on RANS simulation was obtained. It seems hard to correctly predict film cooling flows by RANS turbulence models. Furthermore, limited by computational cost, it is not practical to accomplish the simulation on turbulent flow with very high resolution grid system. LES, a turbulence model that resolves time-dependent turbulent dynamic,

---

* Corresponding author at: State Key Laboratory for Strength and Vibration of Mechanical Structures, School of Aerospace, Xi'an Jiaotong University, 28 Xianning West Road, Xi'an, Shaanxi 710049, China.

E-mail address: wangxian@mail.xjtu.edu.cn (X. Wang).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2016.02.089
0017-9310/© 2016 Elsevier Ltd. All rights reserved.

can eliminate the deficiency of RANS models. Tyagi and Acharya [7] made a first attempt to fulfill the study on film cooling flow using LES based on Navier–Stokes equations, and results showed that LES was able to predict the flow field with more reasonable accuracy comparing with the two-equation models. Soon afterwards, many more numerical studies on film cooling have been conducted with LES and reasonable results were obtained. Guo et al. [8] carried out numerical simulations based on LES to study the turbulent flow structure and the vortex dynamics in gas turbine blade film cooling with 4.6–5.8 million grids. Renze et al. [9] investigated the impact of the velocity and density ratio on the turbulent mixing process in gas turbine blade film cooling using LES with 5.65 million grids. LES of leading edge film cooling was conducted by Rozati and Tafti [10] with about 9.6 million grids to analyze the flow structures, effectiveness and heat transfer coefficient. In 2013, Andrew et al. [11] performed a blind LES of film cooling with 88.7 million meshes. This is by far the most massive computational grid system in our minds, and it was run on 256 processors (8 nodes of quad-processor eight-core 2.0 GHz AMD Opteron 6128), 512 GB ram, and a 20 Gb/s infiniband interconnect. It took about 2 months to accomplish this simulation and a computational performance of 3.5 MLUPS was obtained. The relatively fine results could not be obtained by the aforementioned computations based on LES because of the restricted computational grid number which is caused by the limited ability of central processing unit (CPU).

In recent years, lattice Boltzmann method (LBM), one of the meso-scale methods, has developed fast and been regarded as a promising alternative for simulation on fluid flows with complex physics. This is because of its advantages, such as easy implementation of boundary conditions, easy programming, and fully parallel algorithms [12]. In particular, its fully parallel algorithm makes it match perfectly with graphic processing unit (GPU). GPU has become popular in application of CFD due to its high performance of floating-point arithmetic operation, wide memory bandwidth, and better programmability [13]. According to our experience, solving incompressible Navier–Stokes equations to simulate fluid flow with the marker and cell (MAC) solver on single GPU is 30–40 times faster than the heavily optimized CPU-based implementations. And the calculations based LBE-GPU can even obtain more than 100 speedups [13–15].

Aiming at capturing the fine structures and the interactions among them to reveal the mixing mechanism in film cooling, we attempt to reappear this complex process with a high-resolution computational grid system using hybrid thermal lattice Boltzmann method (HTLBM), an extension of LBM, and multiple graphic processing units (multi-GPUs).

In this study, we perform the large-scaled simulation of single-jet film cooling with the standard Smagorinsky subgrid-stress (SGS) model using our in-house code based on HTLBM with CUDA-MPI [14,15]. The simulations are fulfilled in a parallel way on 9 K20M GPUs with the maximum grid number of 1.65×10^8.

The paper is organized as follows. After a brief description of film cooling configuration in the second section, the solution methodology is presented in the third section. Subsequently, in the fourth section, the validation of our in-house code is presented, the importance of large-scaled grid system on film-cooling investigation is emphasized, and the partition in the downstream region of jet hole is discussed in detail. Then, the blowing-ratio effect on unsteady characteristics and cooling efficiency is studied. Finally, conclusions are made in the last section.

2. Film cooling configuration

Fig. 1 shows the flow configuration of the simulation. A single cylindrical hole inclined at 30° is drilled on the bottom wall. The center of hole locates at the site $10D$ far from the inlet of hot crossflow. D is the hole diameter. The solution domain is $L_x = 35D,\ L_y = 3D$, and $L_z = 10D$. Here, the origin of coordinate system is situated at the center of jet hole, with the x-, y-, and z-axes representing the streamwise, lateral, and wall-normal directions, respectively. 54 grid points are arranged for the length of $1D$, resulting in the total mesh number is 1.65×10^8. The Reynolds number is set as $\mathrm{Re} = (\rho u_\infty D)/\nu = 4000$. The blowing ratio $R = \rho_j u_j / \rho_\infty u_\infty$ ranges from 0.2 to 0.8, where the crossflow density is assumed to be uniform with that of jet flow. The temperature ratio T_j/T_∞ is 0.5. T_∞ and T_j represent the crossflow temperature and jet temperature, respectively. The non-dimensional temperature $\theta = (T_{aw} - T_j)/(T_\infty - T_j)$ and film cooling effectiveness $\eta = (T_\infty - T_{aw})/(T_\infty - T_j)$ are defined to evaluate cooling performance, where T_{aw} is the temperature of the adiabatic bottom wall.

For the inlet crossflow velocity boundary condition, a 1/7 power law velocity profile, $u(z) = \begin{cases} u_\infty (z/\delta)^{\frac{1}{7}},\ 0 \leqslant z \leqslant \delta \\ u_\infty,\ z \geqslant \delta \end{cases}$, is adopted. The turbulent boundary layer thickness δ is assumed to be $2.0D$. The jet-exit velocities are given by the velocity distribution of developed turbulent flow in tube. Periodic boundaries are applied in spanwise direction. In addition, the bottom wall is assumed to be adiabatic and no-slip.

3. Solution methodology

Thermal lattice Boltzmann method (TLBM) is one of the most common methods used to simulate the fluid flow and heat transfer. However, this method encounters some numerical instabilities when simulating turbulent flow. On the other hand, for large-scaled computing, the disadvantage of TLBM, memory consuming, becomes more evident. In this study, we apply HTLBM to carry out the research on flow and heat transfer process in film cooling. HTLBM is the extension of LBM which explicitly couples an athermal lattice Boltzmann equation (LBE) scheme for the flow field and the diffusion–advection equation for temperature field.

3.1. Lattice Boltzmann equation (LBE) with Smagorinsky subgrid scale model

For flow field, single-relaxation-time lattice Boltzmann scheme (SRT-LBM) is used. Adopting Boltzmann-BGK approximation, the further discrete form in physical space $\mathbf{x}$ and time t of LBE is [16,17]:

$$f_i(\mathbf{x} + \mathbf{e}_i \delta t, t + \delta t) - f_i(\mathbf{x}, t) = -\frac{1}{\tau}[f_i(\mathbf{x}, t) - f_i^{eq}(\mathbf{x}, t)]\ (i = 1, 2, \ldots, N) \tag{1}$$

where, f_i and f_i^{eq} is the particle velocity distribution function and local equilibrium distribution, respectively. $\mathbf{e}_i$ is the particle velocity in the ith direction, λ is the relaxation time, and N is the number of velocities. For 2D model, 9-velocity LBM is used extensively, that is D2Q9. For 3D model, there are several cubic lattice models, such as D3Q13, D3Q15, D3Q19 and D3Q27 ($N = 13, 15, 19$ or 27).

Eq. (1) is the well known LBGK model. Here, $\tau = \lambda/\delta t$ is the non-dimensional relaxation time. The viscosity in the macroscopic Navier–Stokes equation can be derived from Eq. (1) as:

$$\nu = \left(\tau - \frac{1}{2}\right)\delta t \tag{2}$$

Eq. (1) is usually solved with its standard form by assuming $\delta t = 1$ according to the following two steps [18]:

$$\text{Collision step}:\quad \tilde{f}_i(\mathbf{x}, t) = f_i(\mathbf{x}, t) - \frac{1}{\tau}[f_i(\mathbf{x}, t) - f_i^{eq}(\mathbf{x}, t)] \tag{3}$$

$$\text{Streaming step}:\quad f_i(\mathbf{x} + \mathbf{e}_i, t + 1) = \tilde{f}_i(\mathbf{x}, t) \tag{4}$$

Download English Version:

https://daneshyari.com/en/article/7055811

Download Persian Version:

https://daneshyari.com/article/7055811

Daneshyari.com