International Communications in Heat and Mass Transfer 87 (2017) 72–83

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

Analysis on the mechanism of evolutionary process of counter-rotating vortex pair in film cooling based on hybrid thermal lattice Boltzmann method

Yanqin Shangguan, Xian Wang*, Hu Zhang, Yueming Li

State Key Laboratory for Strength and Vibration of Mechanical Structures, 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

A R T I C L E I N F O

Keywords:
Large-scaled simulation
Hybrid thermal lattice Boltzmann method
Multiple graphic processing units
Film cooling
Counter-rotating vortex pair
Mechanism of evolutionary process

A B S T R A C T

Massively parallel simulation applying multiple graphic processing units (multi-GPUs) is carried out to perform a deep going investigation on the counter-rotating vortex pair (CVP) in single-jet film cooling based on hybrid thermal lattice Boltzmann method (HTLBM) and large eddy simulation (LES). The mechanism of evolutionary process of CVP and its influence on the Reynolds shear stress and cooling performance are studied in detail. In the simulation, the blowing ratio of coolant jet is kept to $BR = 0.5$ and the inclined angle is $\alpha = 30^{\circ}$. The Reynolds number based on the crossflow velocity and diameter of the jet hole is $\mathrm{Re} = 4000$. The secondary anti-kidney vortices, tertiary kidney vortices and quaternary anti-kidney vortices are captured, which increase the spanwise coolant-film coverage on the surface of the bottom wall. The size of the primary CVP significantly influences the distribution of Reynolds shear stress R_{uv} and R_{uw}. The vortex strength of the primary CVP and the characteristics of minor counter-rotating vortices mainly impact on the streamwise and spanwise distribution of film cooling effectiveness, respectively.

1. Introduction

The turbine inlet temperature continues to rise to further improve the output and efficiency of modern gas turbine, which requires the improvement of turbine cooling techniques. Film cooling, as one of the most important cooling methods, is widely used to protect the components from being destroyed by hot crossflow [1]. The mixing mechanism of film cooling is highly complex and a large amount of vortical structures are formed in the mixing process. The evolution of vortical structures, which has a great effect on coolant jet behavior and cooling performance, is one of the most important physical problems associated with the film cooling. Among these, one of the most important vortical structures is counter-rotating vortex pair (CVP) which is often referred to as kidney-shaped vortex. It determines the dominant features of the velocity and vorticity fields, and its dynamics are mainly responsible for mixing and heat transfer process [2]. Therefore, the detailed understanding of the CVP's evolution and its impact on flow and heat transfer is necessary to effectively design aerodynamic and structural parameters in film cooling.

Over the last five decades, many researchers have discussed CVP. However, most of them paid much attention to the formation of CVP, and studies on CVP to understand its evolution and its enhancement on the lift-off of the coolant jet are rare. Scorer [3] is the first who paid attention to the presence of CVP in crosswind in 1958. He proposed that CVP was formed from the roll-up of the vorticity at the sides of the round orifice only. Broadwell and Breidenthal [4] analyzed the flow field induced by a jet in incompressible crossflow and believed that CVP arised from the impulse associated with the jet. Some detailed experimental studies in the near field of jet in crossflow suggested that CVP was formed by the vortex sheet emanating from the jet hole [5–7]. The development of large-scaled structures of a jet normal to crossflow was studied experimentally by Lim et al. [8]. Their results showed that the presence of CVP inhibited the formation of the vortex rings. Along with rapid development of computer technology, the application of numerical simulation became prevalent in the early 21st century. Muppidi and Mahesh [9,10] performed direct numerical simulation (DNS) on jets in crossflow and found that the formation of CVP was related to the downstream region with adverse pressure. A fundamental, mechanistic understanding of the structure and evolution of vorticity in the transverse jet was sought by Marzouk and Ghoniem [11] with a numerical method named vortex method in 2007. They believed that it was the deformation and periodic roll-up of the shear layer that contributed to

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

http://dx.doi.org/10.1016/j.icheatmasstransfer.2017.06.004

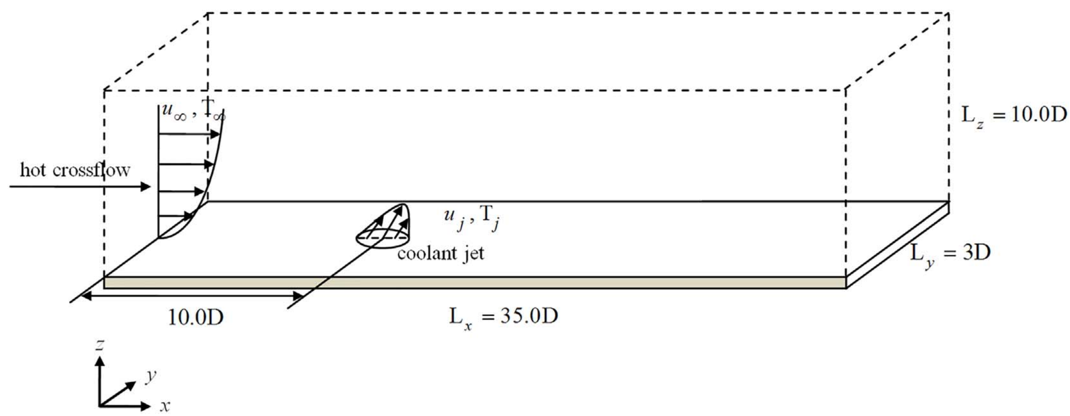

Fig. 1. Flow configuration.

the appearance of CVP. In 2011, Schlegel et al. [12] sought mechanistic understanding of vorticity dynamics in transverse jets using high-resolution 3-D vortex simulations. They found CVP in the first few diameters above the jet exit was resulted primarily from the entrainment of wall boundary layer vorticity via the tornado-like wall-normal wake vortices.

In order to obtain the exact developing process of CVP, the adequate fine quantitative results are necessary. Thus, high-resolution numerical simulations are indispensable. Due to the low performance of central processing unit (CPU), previous computations were performed with limited grid number and could not obtain the fine-scaled results. Miao and Wu [13] carried out numerical simulations to study the effects of blowing ratio and hole shape on the distributions of flow field and adiabatic film cooling effectiveness over a flat plate collocated with two rows of injection holes in staggered-hole arrangement with about 1.0 million grids. There showed a large difference between the simulated streamwise distributions of spanwise averaged film cooling effectiveness and the experimental results of Ai et al. [14]. Large eddy simulation (LES) of leading edge film cooling was conducted by Rozati and Tafti [15] with about 9.6 million grids to analyze the flow structures, effectiveness and heat transfer coefficient. However, there still existed a discrepancy between the simulated and experimental results. In 2013, Andrew et al. [16] 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. However, the simulated film cooling effectiveness still did not coincide well with the experimental data. Moreover, their simulation 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.

In recent years, graphic processing unit (GPU) has become popular in application of computational fluid dynamics (CFD) due to its high performance of floating-point arithmetic operation, wide memory bandwidth, and better programmability [17]. On the other hand, 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. Its remarkable advantages are easy implementation of boundary conditions, easy programming, and fully parallel algorithms [18]. In particular, its fully parallel algorithm makes it well suited for GPU usage. 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. While calculations based LBE-GPU can even obtain more than 100 speedups [17,19,20].

Aiming at capturing the fine structures and obtaining the adequate fine quantitative results to reveal the dynamics of CVP, we attempt to reproduce 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 work, we perform 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 [19,20]. The CVP evolutionary mechanism and its influence on cooling performance associated with single-jet film cooling are concerned. The simulations are performed in a parallel way on 12 K20M GPUs with the maximum grid number of 1.65×10^8.

The paper is organized as follows. After a brief description of computational configuration in the second section, the solution methodology is presented in the third section. Subsequently, in the fourth section, the dynamics and evolution of CVP are shown and its characteristics are discussed in detail. The influence of CVP on characteristics of Reynolds stress and cooling performance is studied. Finally, conclusions are made in the last section.

2. Computational 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 the jet hole is located 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 the 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 1.68×10^8. The Reynolds number is set as $\mathrm{Re} = (\rho u_\infty D)/\nu = 4000$. The blowing ratio $R = \rho_j u_j/\rho_\infty u_\infty$ is 0.5, 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, $u(z) = \begin{cases} u_\infty (z/\delta)^{1/7}, & 0 \le z \le \delta \\ u_\infty, & z \ge \delta \end{cases}$, is adopted. The turbulent boundary layer thickness δ is assumed to be 2.0D. The friction velocity is calculated as $u_\tau = u_e\sqrt{\overline{C_f}/2}$. The velocity of the first element near the wall is $u_e = u_\infty [1/(2D)]^{1/7} = 0.512u_\infty$, and the skin-friction coefficient of the bottom wall $\overline{C_f}$ is approximately $(2 \times 0.037)/\mathrm{Re}^{1/5}$, which results in that the friction velocity is equal to $4.3 \times 10^{-2} u_\infty$. The jet-exit velocities are given by the velocity distribution of fully 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