International Journal of Heat and Mass Transfer 86 (2015) 90–100

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Numerical boundary layer investigations of transpiration-cooled turbulent channel flow

W. Dahmen [a], S. Müller [a,*], M. Rom [a], S. Schweikert [b], M. Selzer [c], J. von Wolfersdorf [b]

[a] Institut für Geometrie und Praktische Mathematik, RWTH Aachen University, Templergraben 55, 52056 Aachen, Germany
[b] Institut für Thermodynamik der Luft- u. Raumfahrt, University Stuttgart, Pfaffenwaldring 31, 70569 Stuttgart, Germany
[c] Institut für Bauweisen und Strukturtechnologie, DLR Stuttgart, Pfaffenwaldring 38-40, 70569 Stuttgart, Germany

ARTICLE INFO

Article history:
Received 13 November 2014
Received in revised form 24 February 2015
Accepted 25 February 2015
Available online 14 March 2015

Keywords:
Transpiration cooling
Numerical two-domain coupling
Darcy–Forchheimer
Compressible RANS
Turbulent boundary layer
Skin friction reduction

ABSTRACT

Transpiration cooling is an active cooling technique able to reduce thermal loads of applications exposed to extreme operation conditions. A coolant is driven through a porous material by a pressure difference between the coolant reservoir and the hot gas flow. The present study numerically investigates the influence of the cooling gas injection on the temperature, the velocity and the local skin friction in the boundary layer of a subsonic turbulent hot gas channel flow. Here, the hot gas in the channel flows over a cooled porous ceramic matrix composite (CMC) material. Separate solvers are used for the hot gas flow and the porous medium flow, respectively. These are applied alternately and coupled to each other by boundary conditions imposed at the interface. The simulation results are compared with experimental data to validate the two solvers as well as the coupling and to provide complementary insight into the effects of the cooling which cannot be assessed from experimental measurements.

1. Introduction

Transpiration cooling is a promising concept for actively cooled structures under high thermal loads. It is based on the injection of coolant into a hot gas boundary layer through a porous material. The injection is driven by a pressure difference between the hot gas and the coolant side. In general, transpiration cooling exhibits two different cooling effects: (i) the material internal heat exchange between solid and coolant, resulting in heat transfer out of the porous wall and (ii) the formation of a protective layer with lower temperatures, resulting in decreased wall heat fluxes.

The superior characteristics of transpiration cooling were already shown by Eckert and Livingood [1]. With respect to possible applications however, the lack of proper materials led to minor interest in this cooling technique until permeable ceramics became available. Nowadays, the combination of transpiration cooling with permeable high temperature fiber ceramics promises to be a powerful thermal protection system. Hence, the DLR (German Aerospace Center) is actively developing transpiration cooled CMC combustion chambers since more than a decade. The successful implementation of this technology was shown in several tests of H2/O2 combustion chambers operating at representative upper stage conditions. Recent developments of this technology are for example given by Ortelt et al. [2] or Herbertz and Selzer [3].

Flow in porous structures was studied for different material properties of two-dimensional homogeneous packed beds, e.g., by Amiri and Vafai [4]. The effect of different boundary condition definitions on such a model was investigated by Alazmi and Vafai in [5]. Extending these studies to a two-layer porous model with different properties was done via 1D-analytical solutions by e.g., von Wolfersdorf [6] and more recently with a numerical approach for a 2D-case by Liu et al. [7]. For accurately capturing the two above-mentioned effects of transpiration cooling, one has to define a coupling between coolant flow and cross flow. Model based coupling without detailed numerical simulation of the cross flow was presented for a 1D-case by Langener et al. [8] and for a 2D-case by Böhrk et al. [9]. Numerical simulations of hot gas flows exposed to transpiration cooling were conducted by Jiang et al. [10] and more recently by Liu et al. [11]. Jiang et al. [10] used fixed boundary conditions to determine the effect of coolant injection into the cross flow. Liu et al. [11] applied combined simulations of hot gas and porous flow to show the qualitative influence of transpiration cooling on the boundary layer of the cross flow. Bellettre et al. [12] measured boundary layer profiles with injection and developed a simplified model in which the porous medium is represented by a succession of single holes. The same model was applied by Bataille et al. [13] and later by

* Corresponding author.

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.02.075
0017-9310/

Brillant et al. [14], simulating an experimental reference case [15]. Further experimental investigations of hot gas flow exposed to transpiration cooling were conducted e.g., by Meinert et al. [16] or Schweikert et al. [17]. Both authors used measured boundary layer profiles over transpiration-cooled walls to assess skin friction and heat flux reduction effects.

In this paper, we focus on the numerical description of transpiration cooling as a coupled problem. In this context, the interface between hot gas and coolant flow is independent of restrictive boundary conditions or model approaches to describe transpiration cooling effects. In doing so, one is able to draw conclusions on the interaction of coolant injection and cross flow which do not have to be related to model assumptions or simplifications for this region of interest.

The simulations are carried out using separate solvers for the hot gas flow and for the porous medium flow, respectively. These are directly coupled to each other by alternately exchanging data at the interface where the coolant enters the hot gas flow. Since we are only interested in steady state solutions, the coupling is realized in a weak sense, i.e., we use a two-domain approach: both solvers are applied alternately where in each iteration the respective solver is converged to a steady state with respect to the boundary conditions at the interface provided by the solution of the other solver. This process is continued until no further changes in both solutions occur. One of the main challenges of this approach is the development of appropriate boundary conditions for the coupling at the interface.

From a numerical point of view, a monolithic approach would be most convenient since it avoids the difficult setup of coupling conditions. This would require a unified model, see for instance [18]. However, to realize such a unified model in an already existing solver causes significant changes, involving a time-consuming implementation process. On the other hand, a two-domain approach allows for the coupling of different models corresponding to different types of partial differential equations, and the application of adjusted discretizations. Hence, since we use separate solvers, we obtain a higher flexibility regarding the coupling with other solvers. Moreover, we can develop optimal models for coupling or easily exchange single coupling conditions, e.g., to investigate the influence of locally varying coolant mass fluxes.

Concerning the turbulent hot gas channel flow, we use the adaptive parallel solver Quadflow [19] which solves the compressible Reynolds-averaged Navier–Stokes equations. The core ingredients of Quadflow are: (i) the flow solver concept based on a finite volume discretization, (ii) the grid adaptation concept based on wavelet techniques and (iii) the grid generator based on B-spline mappings. A variety of turbulence models is available from which we choose Wilcox's two-equation k–ω model with a modification for mass injection [20].

The porous medium flow is modeled by the continuity equation, the Darcy–Forchheimer equation and two temperature equations for both fluid and solid material. This model is discretized by a finite element scheme using the deal.II library [21]. In the following, we will refer to the two solvers as the flow solver and the porous medium solver, respectively.

First results for the validation of the two solvers and, in particular, of the coupling are reported in [22–24]. In addition to the two-dimensional simulations with air as cooling gas presented in this paper, these former works contain results of three-dimensional simulations and simulations with argon instead of air injection. In this work, we focus on the effect of the cooling gas injection on the boundary layer of the hot gas flow. The comparisons with experimental data from Schweikert et al. [17] include boundary layer profiles for the temperature and the velocity in flow direction as well as the local skin friction reduction induced by the injection.

The present paper is structured as follows: the governing equations for modeling the turbulent hot gas channel flow and the porous medium flow as well as suitable boundary conditions are discussed in Sections 2 and 3, respectively. In Section 4, the numerical methods for solving the coupled problem are described, addressing the discretizations for both solvers and the coupling strategy. Numerical results for a coupled two-dimensional simulation are presented and compared with experimental data in Section 5. A summary of the main results and an outlook on future work in Section 6 conclude the paper.

2. Physical modeling: hot gas flow

The turbulent hot gas channel flow can be described by the compressible Reynolds-averaged Navier–Stokes equations composed of equations representing the conservation of mass, momentum and total energy. These have to be complemented by a turbulence model.

2.1. Governing equations

The Reynolds-averaged Navier–Stokes equations (RANS) are obtained by applying the Reynolds averaging

$$f(t,\mathbf{x}) = \bar{f}(t,\mathbf{x}) + f'(t,\mathbf{x}) \quad \text{with} \quad \bar{f}(t,\mathbf{x}) := \lim_{\Delta\to\infty} \frac{1}{\Delta}\int_t^{t+\Delta} f(\tau,\mathbf{x})d\tau \tag{1}$$

to each flow quantity f of the Navier–Stokes equations. In the compressible case, the resulting equations have a rather complex form due to fluctuations in the density ρ. Hence, the density averaging

$$f = \tilde{f} + f'' \quad \text{with} \quad \tilde{f} := \frac{\overline{\rho f}}{\bar{\rho}}, \tag{2}$$

which is called Favre averaging, is applied additionally. The turbulence quantities are modeled by Wilcox's two-equation k–ω model in its latest formulation [20], leading to two additional equations for the turbulence kinetic energy $\tilde{k}$ and the turbulent dissipation $\tilde{\omega}$, respectively. Both are mass-specific values and Favre-averaged. Overall, the set of equations in dimensional form using the Einstein summation convention reads

$$\frac{\partial\bar{\rho}}{\partial t} + \frac{\partial(\bar{\rho}\tilde{v}_j)}{\partial x_j} = 0, \tag{3}$$

$$\frac{\partial(\bar{\rho}\tilde{v}_i)}{\partial t} + \frac{\partial(\bar{\rho}\tilde{v}_i\tilde{v}_j)}{\partial x_j} = -\frac{\partial\bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\bar{\tau}_{ij} + \bar{\rho}\tilde{R}_{ij}\right), \tag{4}$$

$$\frac{\partial(\bar{\rho}\tilde{E})}{\partial t} + \frac{\partial(\tilde{v}_j(\bar{\rho}\tilde{E}+\bar{p}))}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\tilde{v}_i\left(\bar{\tau}_{ij} + \bar{\rho}\tilde{R}_{ij}\right) - \left(\bar{q}_j + \bar{q}_j^t\right) + \overline{v_i''\tau_{ij}} - \frac{1}{2}\overline{\rho v_i'' v_i'' v_j''}\right), \tag{5}$$

$$\frac{\partial(\bar{\rho}\tilde{k})}{\partial t} + \frac{\partial(\bar{\rho}\tilde{v}_j\tilde{k})}{\partial x_j} = \bar{\rho}\tilde{R}_{ij}\frac{\partial\tilde{v}_i}{\partial x_j} - \beta^*\bar{\rho}\tilde{k}\tilde{\omega} + \frac{\partial}{\partial x_j}\left(\left(\mu + \sigma^*\frac{\bar{\rho}\tilde{k}}{\tilde{\omega}}\right)\frac{\partial\tilde{k}}{\partial x_j}\right), \tag{6}$$

$$\frac{\partial(\bar{\rho}\tilde{\omega})}{\partial t} + \frac{\partial(\bar{\rho}\tilde{v}_j\tilde{\omega})}{\partial x_j} = \alpha\frac{\tilde{\omega}}{\tilde{k}}\bar{\rho}\tilde{R}_{ij}\frac{\partial\tilde{v}_i}{\partial x_j} - \beta\bar{\rho}\tilde{\omega}^2 + \sigma_d\frac{\bar{\rho}}{\tilde{\omega}}\frac{\partial\tilde{k}}{\partial x_j}\frac{\partial\tilde{\omega}}{\partial x_j} + \frac{\partial}{\partial x_j}\left(\left(\mu + \sigma\frac{\bar{\rho}\tilde{k}}{\tilde{\omega}}\right)\frac{\partial\tilde{\omega}}{\partial x_j}\right). \tag{7}$$