Numerical Investigation on Radiation Effect in Transpiration Cooling

Kang Qian¹, Taolue Liu¹, Fei He¹*, Meng Wang¹, Longsheng Tang² and Jianxing Zhou²

¹ Department of Thermal Science and Energy Engineering, University of Science and Technology of China, Jinzhai Road 96, Hefei 230027, China
² Beijing Aerospace Technology Institute, Beijing 100074, China
Email: hefeihe@ustc.edu.cn

Abstract. This paper proposed a numerical strategy which could achieve the coupled modeling and solving of transpiration cooling with external high-temperature gas flow and especially take the radiation effect into account. Based on the numerical strategy, the heat and mass transfer characteristics of the transpiration cooling in a high-temperature gas channel were studied, and the radiation effect and corresponding influence factors were analyzed. The results indicated that the radiative heat flux takes an important role in the heat transfer between the transpiration cooling and external high-temperature gas flow which may reach 40% under the operating condition considered in this work, and the radiation absorption from the coolant is more obvious near the downstream wall. As the wall emissivity increases, the radiation heat transfer in the downstream area of the porous wall is enhanced significantly and thereby the wall temperature there increases, as the result, the uniformity of the temperature distribution on the whole porous wall is improved to some extent.

Keywords. Transpiration cooling, radiation, numerical.

Nomenclature

| Symbol | Description |
|--------|-------------|
| T      | temperature |
| I      | radiation intensity |
| K      | permeability |
| F      | inertia coefficient |
| J      | diffusive flux of species |
| k      | thermal conductivity |
| α      | specific area |
| ε      | porosity    |
| s      | solid       |
| f      | fluid       |
| eff    | effective   |
| ms     | mainstream  |

1. Introduction

Due to the high efficiency and uniform performance, transpiration cooling has been regarded as one of the most promising thermal protection techniques for the structures suffering extremely thermal environment, such as the leading edge and combustion chamber of high-speed vehicles. There have been many researches on the mechanism, influencing factors and configuration optimization of transpiration cooling. However, the coupled modeling and solving of the transpiration cooling with external thermal environment and gas flow are still insufficient, and especially the radiation effect from the external field is often excluded. As the result, the transpiration cooling performance is always overestimated under the fuel gas conditions where the radiation effect from the high-temperature gaseous mixture can’t be ignored.
Recently, some researchers attempt to study the radiation effect in transpiration cooling by simplified method. Zhang [1] studied the effect of external radiation on a one-dimensional transpiration cooling problem by introducing a radiative thermal conductivity in the solid energy equation and calculating the radiative heat flux at the boundary through approximate radiation heat transfer equation, so there was no radiation field in his research. Jiang [2] studied the heat transfer in the rectangle channel with transpiration cooling and obtained the radiation heat transfer equations between the surfaces based on the view factor, but his work didn’t include the radiation effect from the participating gaseous media. The above treatments of radiation effect are limited because the radiation from the surfaces and participating gases are mutually influenced and should be considered synchronously.

In this paper, a numerical strategy which could calculate the radiation heat transfer between transpiration cooling and external flow field has been proposed, and based on the proposed method, the heat and mass transfer characteristics of the transpiration cooling in a high-temperature gas channel were studied.

2. Modelling and Numerical Method
The physical model considered in this paper is sketched in figure 1(a). Fuel gas mixture after combustion flows across the upper surface of the porous channel wall, and heats the wall through convective and radiative heat transfer. Coolant is injected into pores from the bottom of the wall, flows and absorbs the heat stored in the porous matrix, and forms a protective film on the surface exposed to the hot gas after flowing out.

2.1. Mathematical Models
Different models are respectively developed to describe the fluid flow and heat transfer processes in the mainstream region and porous region.

2.1.1. Mainstream Region. For the steady state mainstream, the conservation equations concerned, i.e. the mass, momentum, species transport, energy, and radiative transfer equations are listed as follows:

\[ \nabla \cdot (\rho u) = 0 \]  \hspace{0.5cm} (1)
\[ \nabla \cdot (\rho u u) = -\nabla P + \nabla \cdot \tau + \frac{\partial}{\partial x_j}(-\rho \overline{uu}_j) \]  \hspace{0.5cm} (2)
\[ \nabla \cdot (\rho u Y_i) = -\nabla \cdot J_i \]  \hspace{0.5cm} (3)
\[ \nabla \cdot (\rho u (h + \frac{u^2}{2})) = \nabla \cdot (k_{eff} T - \sum_i h_i J_i + \tau_{eff} \cdot u) + S_h \]  \hspace{0.5cm} (4)
\[ \nabla \cdot (I(r, s)s) + (\alpha + \alpha_s) I(r, s) = an^2 \frac{\sigma T^4}{\pi} + \frac{\alpha_s}{\pi} \int_0^{4\pi} I(r, s') \phi(s \cdot s') d\Omega' \]  \hspace{0.5cm} (5)
where the Reynold stress is calculated based on the SST \( k - \omega \) turbulent model, and the radiation calculation is based on the DO radiation model.

2.1.2. Porous Region. The mass and momentum equations for the fluid in the porous region are based on the volume-averaged method and Brinkman-Forchheimer extended Darcy’s law, and expressed as:

\[ \nabla \cdot (\rho_f u) = 0 \]  \hspace{0.5cm} (6)
\[ \nabla P = -\frac{\mu}{k} u - \frac{\rho_f}{\sqrt{k}} \overline{|u|} u + \nabla (\mu_{eff} \nabla u) \]  \hspace{0.5cm} (7)

The local thermal non-equilibrium characteristic, i.e., the convective heat transfer between the solid and the fluid in the pore is taken into account, and thus the energy equations for the fluid and the solid matrix are respectively expressed as:

\[ \nabla \cdot (\rho_f U'f) = \nabla \cdot (k_{eff} \nabla T_f) + Q_{sf} \]  \hspace{0.5cm} (8)
\[
\n\nabla \cdot (k_{\text{eff}} \nabla T_s) = Q_{sf}
\]

(9)

The heat transfer between the fluid and the solid is calculated by [3]:

\[
Q_{sf} = \alpha (0.933 \varepsilon^2 - 0.245 \varepsilon + 0.0165) Re_d^{0.8} Pr^{1/3} k_f \left( T_s - T_f \right) / d_p
\]

(10)

2.2. Numerical Strategy

As shown in figure 1(a), the coolant outflow mixes with the mainstream, so besides heat transfer, there is also mass transfer between the mainstream region and the porous region. A numerical strategy that could achieve the coupled solution of the two regions is proposed here by defining the physical parameter transfer at the interface.

The pressure and velocity of fluid in the two regions are consecutive at the interface as suggested by Betchen [4] and Ochoa [5]:

\[
P_p = P_{ms}, \quad u_p = u_{ms}
\]

(11)

An equivalent wall temperature [6] is introduced at the interface due to the temperature deviation between the solid and fluid:

\[
T_{ms} = \varepsilon T_f + (1 - \varepsilon) T_s
\]

(12)

The total heat flux entering the porous region consists of radiation and convection/conduction. The former can be obtained from the calculation of radiation field in the mainstream region, and the latter can be calculated by [7]:

\[
q''_c = -c_p \left( \frac{\mu}{Pr} + \frac{\mu_t}{Pr_t} \right) \nabla T_{ms}
\]

(13)

According to Wang [8], the heat flux distribution between the solid matrix and fluid at the interface can be written as:

\[
q'' = q''_s + q''_f
\]

(14)

\[
\frac{q''_s}{q''_f} = \frac{(1-\varepsilon)k_s}{\varepsilon k_f}
\]

(15)

Numerical simulations were conducted by Ansys Fluent, and UDF was embedded to achieve the coupled calculations between porous and mainstream regions. As shown in figure 1(b), the external flow and radiation fields were calculated in case-1 by receiving the interfacial temperature and velocity distributions from the case-2, the flow and heat transfer inside the porous medium were calculated in case-2 by applying the hot side heat flux and pressure distributions extracted from the case-1. Iterations were carried out through the interfacial data exchange from the results of the two cases and continued until the convergence criteria have been achieved.

![Figure 1](image_url)

**Figure 1.** The schematics of (a) transpiration cooling and (b) coupled solution process.

2.3. Geometry and Test Conditions

The geometrical model is shown in figure 2 with the coordinate system and scales marked on.
The mainstream was a high-temperature gaseous mixture, the coolant was superheated steam, and their components are listed in table 1. In the simulations, the total temperature of the mainstream and the coolant was 2000 K and 500 K respectively, the mass flux of the mainstream was 200 kg/m$^2$s, the mass flow rate of the coolant was 0.03 kg/s, the operating pressure was 0.5 MPa, and the turbulent intensity in the mainstream was 10%.

**Table 1. Components and their mass fraction in the mainstream and coolant.**

| Component | Mainstream | Coolant |
|-----------|------------|---------|
| H$_2$O    | 0.08       |         |
| CO$_2$    | 0.21       |         |
| N$_2$     | 0.71       |         |
| H$_2$O    | 1          |         |

Several cases were designed and in the basic case, the upstream wall and downstream wall were set as adiabatic, all the wall emissivity was set as 0.8. The parameters of the porous medium are listed in table 2. At the cold side, a convective boundary condition was used to account for the heating process of coolant before entering the matrix and a non-dimensional coefficient, $R_c$ [9], was introduced as follows:

$$R_c = h_c/\rho u c_p = 0.25$$  \ (16)

**Table 2. Parameters of the porous medium.**

| $\varepsilon$ | K  | F  | $\alpha$ | $k_s$ |
|----------------|----|----|-----------|-------|
| 0.33           | 8.3E-13m$^1$ | 0.75 | 33500m$^1$ | 20W/mK |

The computational domains were separated into two parts, whose meshes were both generated with ICEM. The grid independence of numerical results was checked by three mesh strategies respectively with the grid number 144000, 210000 and 285000. Under the same boundary conditions, the temperatures on the surface of porous wall were calculated with the three mesh strategies, and the results indicated that the mesh with grid number 210000 was enough for the calculation.

**3. Analyses and Discussions of Numerical Results**

The pressure, steam mass fraction, and temperature distributions in the mainstream and porous regions are presented in figure 3, where the temperature in the porous region on the figure 3(a) is the averaged value ($\varepsilon T_s + (1 - \varepsilon)T_f$). A low-temperature film adherent to the wall could be observed in the mainstream region and it is more apparent near the end of the porous wall ($x \approx 60\,mm$). In the porous region, the fluid temperature is very low at the cold side and increases rapidly as the fluid flowing towards the hot side. The temperature of solid matrix is higher than that of fluid, and the thermal non-equilibrium characteristic is remarkable at the cold side and the hot side. Besides, both the temperature distributions of the fluid and solid at the hot side are not uniform, the temperature difference over the surface exceeds 400K.
To account for the radiation effect of the coolant outflow, the net spherical mean radiation is introduced which is defined as the deviation of the spherical mean absorbed radiation and emitted radiation. A positive value means a radiative sink related to the radiation absorption. The distribution of the net spherical mean radiation in the mainstream region is presented in figure 4. In most area, the value of the net spherical mean radiation is negative, and is particularly lower above the porous wall, indicating that a large amount of radiant heat is dissipated from the gaseous mixture in the mainstream and enters the porous wall. In the film layer formed from the coolant outflow, the value of the net spherical mean radiation is positive, which implies that the coolant outflow has a positive effect on weakening the mainstream-side radiative heat transfer. In addition, as the more coolant flows out and accumulated, the radiation absorption by the coolant outflow is more significant in the downstream region.

The temperature and heat flux distributions on the interface between the two regions are shown in figure 5. At the interface, the solid temperature is a bit higher than that of the fluid and their deviation gets smaller along with the mainstream direction, which more specifically, is 56K at the front and 24K at the end. As shown in figure 5(b), the radiative heat flux takes a large proportion in the total heat flux especially when x>45mm where the ratio of radiative heat flux to the total heat flux reaches 38%~40%.

**Figure 3.** Contours: (a) full temperature (b) porous temperature (c) pressure (d) H$_2$O mass fraction.

**Figure 4.** Net spherical mean radiation.
To analyze the effect of radiant heat transfer on the transpiration cooling quantitatively, two additional cases were calculated where the wall emissivity of 0.4 and no radiation was considered respectively, and the results were compared with the basic case where the wall emissivity was 0.8 and shown in figure 6. Under the same operating condition, the surface temperature of the porous wall increases with the wall emissivity, and the maximum increment reaches 105K at the end of the porous wall. 1) the temperature difference over the porous wall decreases as the wall emissivity increases, which indicates temperature distribution on the surface of channel wall is more uniform at high wall emissivity. All those phenomena can be explained with the heat flux distribution presented in figure 6(b). With the increase of wall emissivity, the heat flux enters the porous wall from the high-temperature mainstream increases and the heat flux distribution is more uniform.

4. Conclusions
The radiation effect from the external flow field on the transpiration cooling has been studied in this paper, and some results could be concluded as follows:

(1) The radiative heat flux takes a large proportion in the total heat flux entering the transpiration cooling structure, which can reach 40% under the operating conditions considered in this work.

(2) The radiation absorption effect from the coolant outflow is more obvious near the downstream wall.

(3) The higher the wall emissivity, the higher the hot-side temperature and the smaller the amplitude of temperature variation on the porous wall.

Acknowledgement
This work is supported by the National Natural Science Foundation of China (contract no. 51806206).
References

[1] Zhang W H, Yi Y, Bai X H and Nakayama A 2020 A local thermal non-equilibrium analysis for convective and radiative heat transfer in gaseous transpiration cooling through a porous wall. *International Journal of Heat and Mass Transfer* **162** 120389.

[2] Jiang P X, Yu L and Ren Z P 2004 Influence of variable thermophysical properties and thermal radiation on convection heat transfer in transpiration cooling *Journal of Aerospace Power* 1000-8055 02-0184-07.

[3] Liu Y Q, Xiong Y B, Jiang P X, Wang Y P and Sun J G 2013 Effects of local geometry and boundary condition variations on transpiration cooling *International Journal of Heat and Mass Transfer* **62** 362-372.

[4] Betchen L, Straatman A G and Thompson B E 2006 A nonequilibrium finite-volume model for conjugate fluid/porous/solid domains *Numerical Heat Transfer, Part A: Applications* **49**(6) 543-565.

[5] Ochoa-Tapia J A and Whitaker S 1998 Momentum jump condition at the boundary between a porous medium and a homogeneous fluid: Inertial effects *Journal of Porous Media* **1**(4) 201-217.

[6] Phanikumar M S and Mahajan R L 2002 Non-Darcy natural convection in high porosity metal foams *International Journal of Heat & Mass Transfer* **45**(18) 3781-3793.

[7] Dahmen W, Müller S, Rom M, Schweikert S, Selzer M and von Wolfersdorf J 2015 Numerical boundary layer investigations of transpiration-cooled turbulent channel flow *International Journal of Heat and Mass Transfer* **86** 90–100.

[8] Wang J H and Shi J X 2008 Discussion of boundary conditions of transpiration cooling problems using analytical solution of LTNE model *Journal of Heat Transfer* **130** 014504-1.

[9] Xiong Y B 2013 *Study of the Mechanism of Flow and Heat Transfer on Supersonic Transpiration Cooling* (Beijing: Tinghua University).