HiSST: International Conference on High-Speed Vehicle Science Technology

26–29 November 2018, Moscow, Russia

Numerical study on a multi-sample transpiration-cooled channel flow

Amadeus Trübsbach [1], *Andreas Schwab* [2], *Christian Kromer* [3], *Markus Selzer* [4]

Abstract

Transpiration cooling is a promising active cooling technique for high-temperature aerospace applications such as rocket combustion chambers or reentry vehicles. It combines a cooling effect inside the material and a cooling film that protects the structure from high heat fluxes. Along the cooled wall, heat flux and pressure vary and consequently, the cooling mass flow needs to be adjusted regarding these changing conditions. In this paper a numerical approach is developed where the cooling mass flow through four porous carbon reinforced carbon (C/C) samples is coupled with a turbulent hot gas channel flow. The results show that the blowing ratio is the most influential parameter for the transpiration cooling system. An increase of the blowing ratio results in a significant reduction of the wall temperature. Besides, the thermal conductivity of the C/C-samples and the Mach number of the hot gas channel flow affect the hot side wall temperature. Furthermore, one of the porous C/C samples is replaced by a copper sample for a comparison with the regenerative cooling method. The result shows that transpiration cooling is more effective than regenerative cooling even for relatively low blowing ratios.

Keywords: *Transpiration Cooling, Coupled CFD*

Nomenclature

Latin
F – Blowing Ratio
k – Turbulent kinetic energy
V – Volume
Greek
ε – Porosity
Θ_s – Non-dimensional solid wall temperature
ω – Specific turbulence dissipation rate
Subscripts
ad – Adiabatic
c – Coolant
h – Hot gas
p – Porous
t – Total

1. Introduction

2. Numerical Approach

The hot gas channel and the porous wall are solved separately and coupled externaly using the commercial computational fluid dynamics (CFD) software ANSYS® CFX. Thereby, a quasi-two-dimensional approach is chosen in order to minimize computational work. Figure 1 shows the general setup of the numerical model that includes the hot gas domain, the four porous C/C samples and the coupling mechanism indicated by the arrows. In the following, a description of each feature is provided.

[1] *Research Engineer, German Aerospace Center, Space Systems Integration, amadeus.truebsbach@dlr.de*

[2] *Research Engineer, Institute of Aerospace Thermodynamics, University of Stuttgart, andreas.schwab@itlr.uni-stuttgart.de*

[3] *Research Assistant, German Aerospace Center, Space Systems Integration, christian.kromer@dlr.de*

[4] *Research Engineer, German Aerospace Center, Space Systems Integration, markus.selzer@dlr.de*

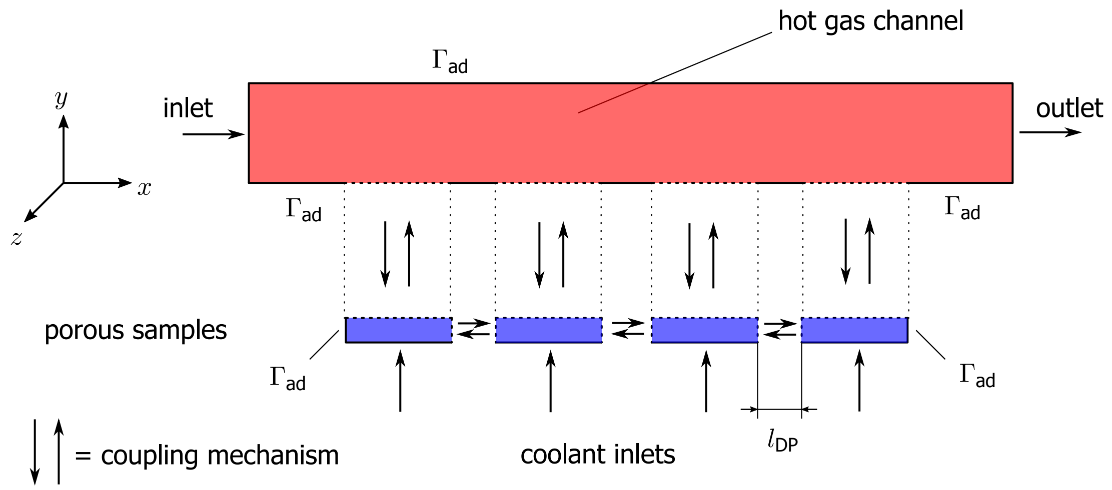

Fig 1. Schematic of numerical model

2.1. Hot gas channel model

The hot gas channel model uses the Reynolds-averaged Navier-Stokes equations (RANS) to approximate the Navier-Stokes equations. As turbulence model, the k-ω SST turbulence model is chosen. To account for possible complex flow behavior in the boundary layer, a near wall treatment is chosen with the size of the first cell set to $y^+ < 1$ above the porous C/C samples. For the inlet, a velocity and temperature distribution of a fully developed turbulent channel flow is set as boundary condition, whereas for the outlet a pressure outlet is chosen.

2.2. Porous model

In the porous C/C samples a laminar flow is observed due to the small dimension of the pores. Therefore, the RANS equations are solved without the turbulent terms, but including the porosity ε that is defined as pore volume V_p per total volume V_t ($\varepsilon = V_p/V_t$). The Darcy-Forchheimer model (see Equation 1) describes the pressure drop across the porous wall and is implemented as source term in the RANS momentum equations:

$$\frac{\partial p}{\partial x_i} = -\frac{\mu}{K_D}\, u_i - \frac{\rho}{K_F}\, |u_i|\, u_i \,. \tag{1}$$

p denotes the pressure, x_i the spatial coordinates, μ the dynamic viscosity. u_i the velocity and ρ the density of the fluid. The material parameters K_D and K_F account for the inertia and viscous forces, respectively. For the energy equation, the two temperature model according to equation 2 and 3 is used:

$$\frac{\partial}{\partial x_i}\left(\rho\, u_i\, h_f\right) = \frac{\partial}{\partial x_i}\left(\varepsilon k_f \frac{\partial T_f}{\partial x_i}\right) + h_v\left(T_s - T_f\right)\,, \tag{2}$$

$$0 = \frac{\partial}{\partial x_i}\left(k_{eff,i}\frac{\partial T_s}{\partial x_i}\right) + h_v\left(T_f - T_s\right)\,. \tag{3}$$

In the above equations h_f is the enthalpy of the fluid, k_f the thermal conductivity of the fluid, k_{eff} the effective thermal conductivity, T_f and T_s the fluid temperature and the solid temperature, respectively, and h_v is the volumetric heat transfer coefficient.

2.3. Coupling mechanism

3. Results

3.1. Equal blowing ratios

Figure 2 shows the non-dimensional solid wall temperature at the hot gas side Θ_s, defined as

$$\Theta_\mathrm{s} = \frac{T_\mathrm{w} - T_\mathrm{c}}{T_\mathrm{g} - T_\mathrm{c}} \tag{4}$$

for various blowing ratios F, defined as

$$F = \frac{\rho_\mathrm{c} u_\mathrm{c}}{\rho_\mathrm{g} u_\mathrm{g}} \,. \tag{5}$$

In this case, all four porous samples are cooled with the same blowing ratio. The dotted perpendicular lines indicate the position of the samples and the gaps between them. The cooling effect becomes already significant for low blowing ratios of $F = 0.05,$%. Blowing ratios of $F > 1$% result in a wall temperature close to the reservoir coolant temperature, i.e. $\Theta_\mathrm{s} = 0$. Between the samples, the temperature rises slightly, because these regions are not transpiration cooled. In the wake region, the coolant film decays due to turbulent mixing and heat conduction and hence rises towards the hot gas channel bulk temperature.

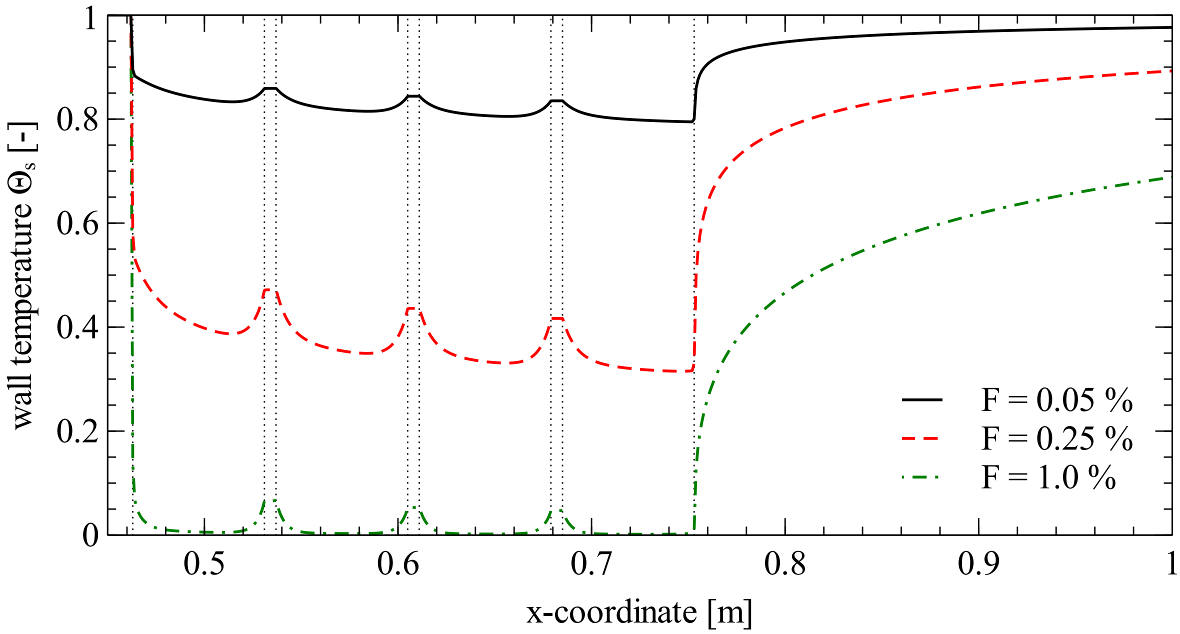

Fig 2. Wall temperature for different blowing ratios

The influence of the blowing ratio on both the wall temperature and the heat flux is additionally shown in Figure 3 where the cooling effect is examined at the center of the first sample.. The heat flux is non-dimensionalised by the maximum heat flux leaving the hot gas channel for a blowing ratio of $F = 0.25$%. When the blowing ratio is increased, the heat flux is increasing as well for low blowing ratios, but decreasing for high blowing ratios. The increase is due to a higher temperature difference between the hot gas and the porous surface temperature. The decrease stems from the developing coolant film, which decreases the temperature gradients and therefore leads to a lower heat flux into the porous samples.

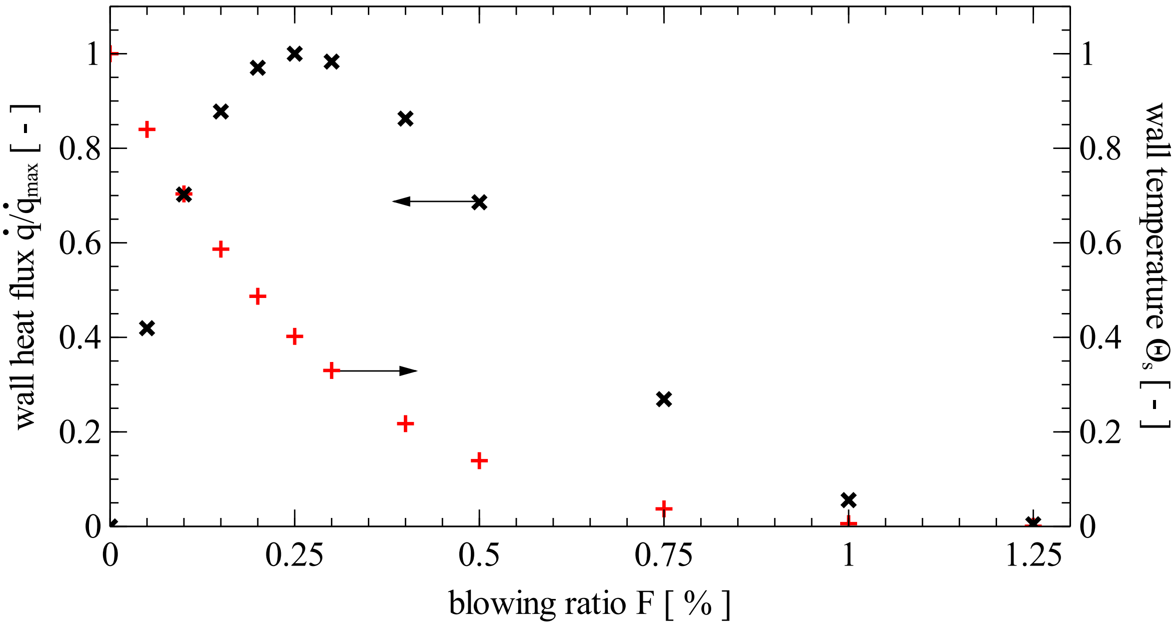

Fig 3. Wall heat flux and temperature over blowing ratio

3.2. Varying blowing ratios

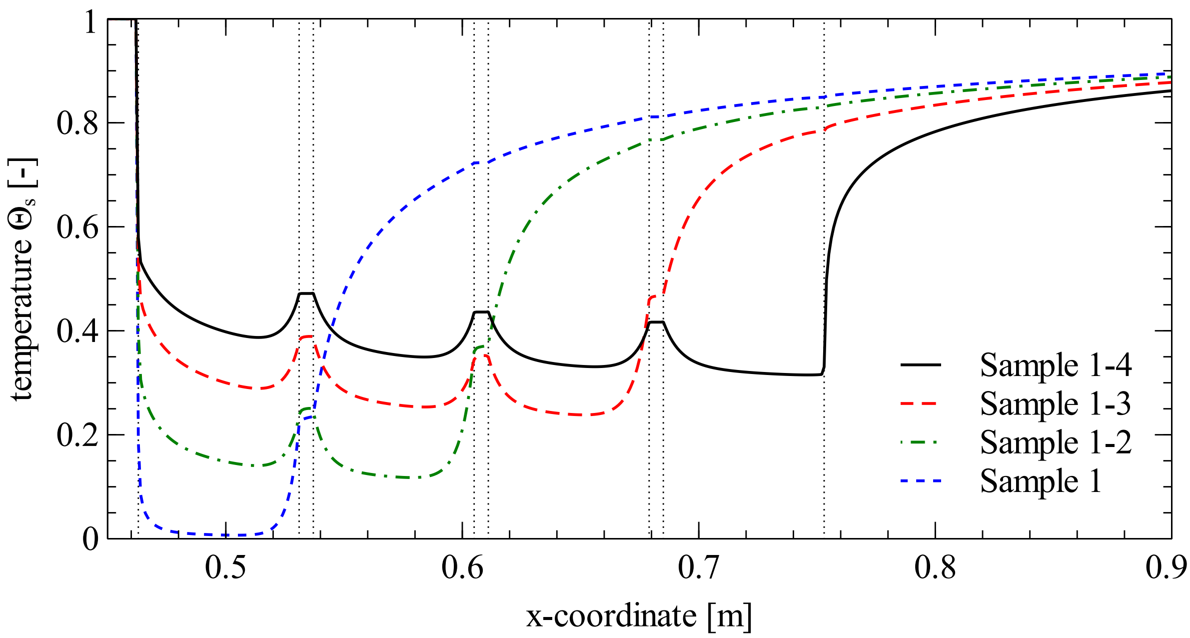

Fig 4. Wall temperature for different number of cooled samples

3.3. Hot gas channel velocity

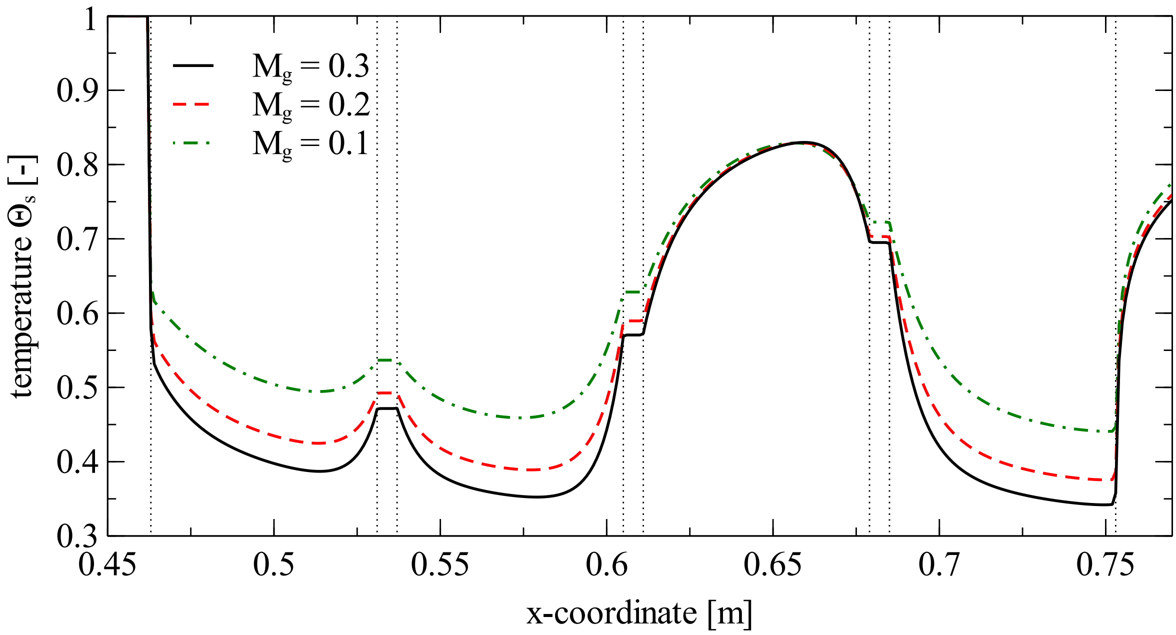

Fig 5. Wall temperature for different cooling methods for the third sample

3.4. Transpiration cooling and regenerative cooling

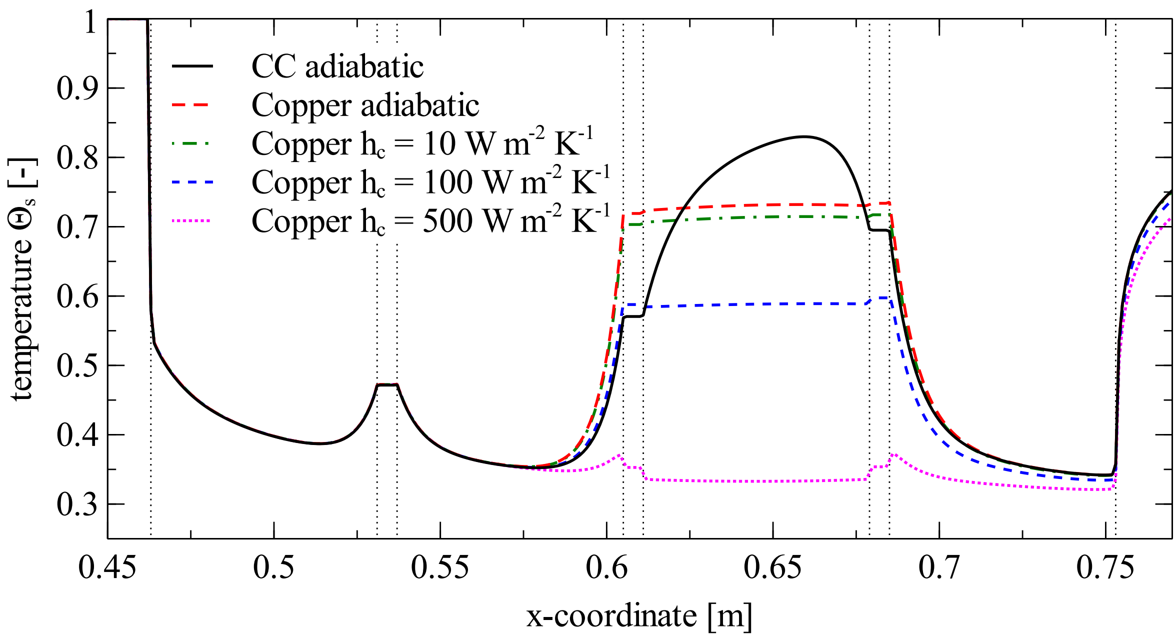

Fig 6. Wall temperature for different cooling methods for the third sample

References