UK Heat Transfer Conference 2021
UKHTC2021

17th UK Heat Transfer Conference (UKHTC2021)
5-7 September 2021, Manchester, UK

HEAT FLUX AND COOLANT CONCENTRATION IN HYPERSONIC TURBULENT FLOW WITH TRANSPIRATION COOLING

Adriano Cerminara

School of Engineering, Computing and Mathematical Sciences, University of Wolverhampton, Telford, TF2 9NT, United Kingdom

1. ABSTRACT

The present contribution shows results from direct numerical simulations (DNS) of a Mach 5 turbulent flow on a flat plate with modelled porous injection at different blowing ratios, representative of a transpiration-cooling system. It is found that the coolant film on the surface is preserved within the calmed region, whereas in the turbulent wedge it decays as a consequence of the wall-normal convective transport. This, in turn, results in the beneficial effect of the blowing ratio on the wall heat flux being relevant within the transitional region, but almost negligible within the turbulent wedge.

1.1 Introduction

The film cooling technique [1] can potentially suppress the heat flux on the surface of hypersonic vehicles, thus providing an efficient solution to the aerodynamic heating problem. Two injection strategies can be considered, namely through localised holes, i.e. effusion cooling [2], and through a transpiring porous material, i.e. transpiration cooling [3]. The latter, in particular, represents a potentially very efficient solution as it provides a more uniform and homogeneous coolant film within the boundary layer. Another strategy is represented by injecting coolant through two-dimensional slots, which helps reducing the 3D effects associated with hole injection and hence delaying transition [4]. Latest studies [5,6] found that porous injection, hence transpiration cooling, performs better then slot injection within the transitional region, due to the enhanced coolant mixing in the boundary layer. However, the effect of different porous structures and injection patterns on the flow features within a hypersonic turbulent boundary layer are still poorly understood, which makes difficult to draw precise engineering estimations on the optimal parameters for maximising the cooling effectiveness. The present study aims at assessing, through DNS simulations, the performance of a porous-injection system in a hypersonic turbulent boundary layer for three different values of the blowing ratio. A periodic function in both the streamwise and spanwise directions has been used to model the coolant blowing profile, mimicking injection through a bed of equally-spaced aligned circular pores.

2. METHODOLOGY, COMPUTATIONAL DOMAIN, SETTINGS AND POROUS INJECTION MODEL

Direct numerical simulations of the full three-dimensional compressible Navier-Stokes equations are performed through the code SBLI, consisting of a 4th -order central finite difference base scheme combined with a 2nd -order Harten-Yee total-variation-diminishing (TVD) shock capturing scheme [8]. The nondimensional system of governing equations includes the continuity equation for the coolant species, which is considered as air. The reader can refer to [5,8] for a detailed definition of the system of governing equations and the associated quantities. A rectangular-box computational domain for a flat plate has been considered, with dimensions $L_x = 300$, $L_y = 30$, $L_z = 48$. The length scales are normalised with respect to the boundary-layer displacement thickness at the inlet boundary. The mesh size in the different directions is $N_x = 1874$, $N_y = 201$, $N_z = 360$, and a grid stretching in the vertical direction towards the wall has been applied in order to accurately resolve the boundary layer. The present grid provides values of $\Delta y+ = 0.34$, $\Delta x+ = 8.2$, $\Delta z+ = 6.8$ at a position ($x = 250$) within the developed turbulent region. These values guarantee DNS resolution in all the directions within the developed turbulent flow region, according to the work of Coleman and Sandberg [9], being the thresholds 1, 15 and 8 for $\Delta y+$, $\Delta x+$, and

*Corresponding Author: A.Cerminara@wlv.ac.uk

$\Delta z+$, respectively. The flow conditions for the present case are the same as those considered in [5]. The flow is initialised with the similarity boundary-layer solution for a Mach 5 flow at a wall temperature of 290 K. The Reynolds number relative to the inlet boundary-layer displacement thickness (representing the characteristic length in our computational domain) is 12600. The wall is set as isothermal. Span-periodic conditions are set at the side boundaries, whereas extrapolation, integral and standard outflow conditions are set at the inlet, top and outlet boundaries, respectively. The injection profile is shown in Figure 1, and for each pore a blowing ratio $F=\rho u_z$ of 0.003, 0.006 and 0.009, has been imposed, respectively, for three different cases. Each pore has a dimensionless diameter of 1.2, and the bed of injecting pores spans the whole domain in the *z*-direction and extends from *x*=55 to *x*=100 (as shown in the top view of the surface in Figure 1, right, for the coolant concentration).

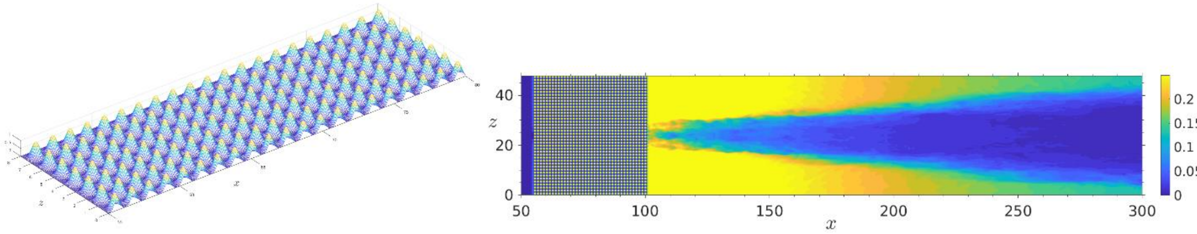

Figure 1: Porous injection profile (left) and instantaneous coolant concentration field (right) at the wall for *F*=0.009

3. RESULTS

Figure 1 (right) reveals that the coolant on the surface is present in relatively high amounts within the calmed flow region, whereas it reduces drastically within the turbulent wedge, due to the convective transport towards the upper layers. Figure 2 shows contours of the temperature within the boundary layer, which highlights details of the turbulent structures developing downstream. Figure 3 shows the streamwise distribution of the time-averaged and spanwise-averaged nondimensional heat flux for the three blowing ratios and the corresponding result for the coolant concentrations on the surface.

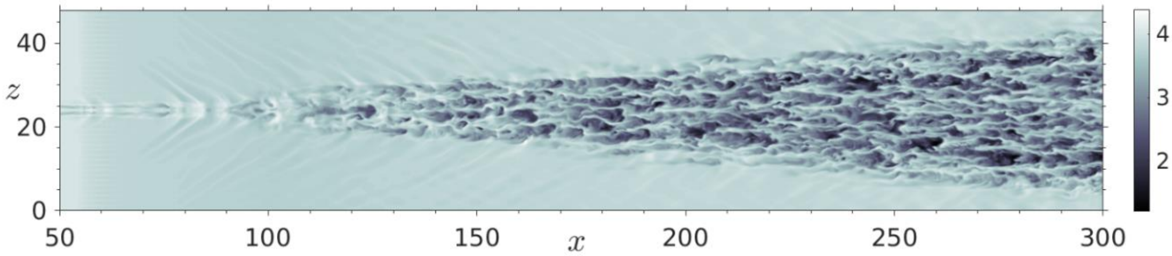

Figure 2: instantaneous temperature field within the boundary layer (*y*=0.44), for *F*=0.009

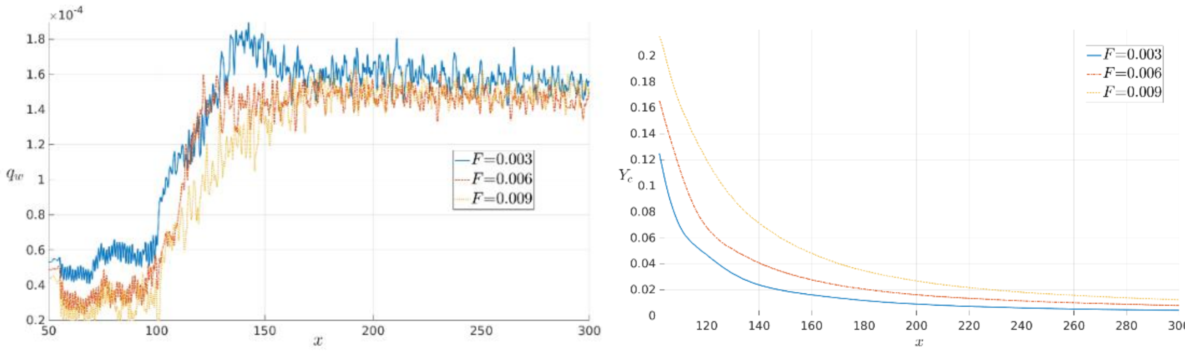

Figure 3: time-averaged and spanwise-averaged nondimensional wall heat flux (left) and coolant concentration on the wall (right) for the different blowing ratios

As can be observed, increasing the blowing ratio from F=0.003 to F=0.006 provides a pronounced reduction of the heat flux peak at the upper limit of the transition region; an increase to F=0.009 provides a further decrease of the heat flux within the transition region and a more gradual variation from the laminar value to the turbulent value. However, consistent with Figure 3 (right), which shows the rapid decay of the coolant concentration on the surface within the turbulent wedge by about an order of magnitude compared to their upstream value, the heat flux profiles for the different blowing ratios collapse to a similar value in the developed turbulent flow region downstream.

4. CONCLUSIONS

The results presented in this work have shown the effect of the coolant blowing ratio on the wall heat flux in a hypersonic turbulent boundary layer, and, contextually, the effect of turbulent transition on the coolant film adjacent to the wall, for transpiration cooling applications. The blowing ratio has been found to produce relevant reduction of the heat flux peak and gradient within the transitional region, but, at the same time, has been observed to be almost ineffective at larger downstream distances from the injection location within the turbulent wedge. The latter, in turn, has been found to induce a significant wall-normal motion of coolant away from the wall, as opposed to the adjacent calmed flow region where the film of coolant in direct contact with the surface develops up to larger downstream distances. This suggests that a continuous bed of injecting pores is needed to preserve the beneficial effects of cooling within a turbulent boundary layer, and that a parametric study aimed at the optimisation of the main porous structure parameters for optimal cooling performance is crucial for the design of reliable thermal protection systems of new-generation hypersonic vehicles.

ACKNOWLEDGEMENTS

Computer time on the UK National Supercomputing Service ARCHER 2 has been provided by the UK Turbulence Consortium (UKTC), chaired by Dr Sylvain Laizet (Imperial College London), under EPSRC (Engineering and Physical Sciences Research Council) grant no. EP/D44073/1, and EP/G06958/1. The present work has been carried out in the scope of the NATO AVT-352 Task Force “Measurement, Modelling and Prediction of Hypersonic Turbulence”, chaired by Dr Carlo Scalo (Purdue University) and Dr Alexander Wagner (German Aerospace Centre, DLR).

REFERENCES

[1] Fitt, A. D., J. R. Ockendon, and T. V. Jones. "Aerodynamics of slot-film cooling: theory and experiment." Journal of fluid mechanics 160 (1985): 15-27.

[2] Baldauf, S., A. Schulz, and S. Wittig. "High-resolution measurements of local heat transfer coefficients from discrete hole film cooling." J. Turbomach. 123, no. 4 (2001): 749-757.

[3] Langener, T., J. V. Wolfersdorf, and J. Steelant. "Experimental investigations on transpiration cooling for scramjet applications using different coolants." AIAA journal 49, no. 7 (2011): 1409-1419.

[4] Keller, Michael A., Markus J. Kloker, and Herbert Olivier. "Influence of cooling-gas properties on film-cooling effectiveness in supersonic flow." Journal of Spacecraft and Rockets 52, no. 5 (2015): 1443-1455.

[5] Cerminara, A., R. Deiterding, and N. Sandham. "A mesoscopic modelling approach for direct numerical simulations of transition to turbulence in hypersonic flow with transpiration cooling." International Journal of Heat and Fluid Flow 86 (2020): 108732.

[6] Cerminara, A., T. Hermann, H. S. Ifti, R. Deiterding, N. Sandham, and M. McGilvray. "Influence of instability modes on cooling performance in hypersonic boundary layer with slot injection." Aerospace Science and Technology 109 (2021): 106409.

[7] Yee, H. C., N. D. Sandham, and M. J. Djomehri. "Low-dissipative high-order shock-capturing methods using characteristic-based filters." Journal of computational physics 150, no. 1 (1999): 199-238.

[8] Cerminara, A., and N. Sandham. "Transition mechanisms in cross-flow-dominated hypersonic flows with free-stream acoustic noise." J Fluid Mech 896 (2020).

[9] Coleman, G. N., and R. D. Sandberg. "A primer on direct numerical simulation of turbulence-methods, procedures and guidelines." (2010).