Transpiration cooling in hypersonic turbulent boundary layer

Adriano Cerminara

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

Abstract

The design of new-generation reliable and efficient hypersonic flight vehicles requires analysis and validation of an effective technique to suppress the intense heat loads that generating on the vehicle surface. The purpose of the present contribution is to study the characteristics of a hypersonic turbulent flow over a porous-injecting wall, representative of a transpiration cooling system, and to analyse the pore-size effect on the coolant performance. Direct numerical simulations (DNS) are carried out for a Mach 5 flow over a flat plate. A porous injection model has been designed which mimics coolant injection from a bed of equally-spaced circular pores. Rapid transition to turbulence is triggered by high-amplitude disturbances imposed on the wall upstream of the porous region. Results show that a turbulent wedge-shaped flow structure generates just downstream of the injection region, which produces a reduction of the surface coolant concentration. The pore size influences flow features and coolant concentration in the laminar region, however has a marginal effect within the turbulent region, where the wall-cooling performance depends predominantly on the fluid dynamics of the turbulent flow. The present work sheds light on the effects of turbulence and pore size on transpiration-cooling characteristics in hypersonic flow, still poorly understood and not in-detail explored in the literature. Results indicate that the turbulent-wedge flow features must be deeper investigated with focus on the coolant redistribution, and that a parametric-study-informed tailored calibration of different porous-injection parameters is vital for controlling the flow features to optimise the cooling performance.

Keywords: Boundary-layer transition, hypersonic flow, transpiration cooling, turbulence

1. Introduction

1.1 Motivation

Hypersonic flight is characterised by extremely high values of the heat flux on the vehicle surface, and consequently surface temperatures that can reach values well above the tolerable limit of the structure. This is due to the aerodynamic heating phenomenon. In addition to that, transition to turbulence contributes to dramatically increasing the heat flux, due to the enhanced mass and energy exchange within the boundary layer. To guarantee the vehicle structure integrity an effective cooling technique must be applied on the surface, which dissipates the intense heat loads and keep the temperature within a tolerable value. Transpiration cooling represents an innovative wall cooling technique with the potential to enable long-duration hypersonic cruise flight in virtue of its high efficiency. However, the mechanisms governing the cooling performance in a turbulent boundary layer, including the way key parameters of the porous structure and injection patterns affect the turbulent flow features are still unknown. This makes difficult any tentative of an ad-hoc design of the porous structure for maximising the cooling effectiveness. It is hence crucial to study the influence of different porous injection parameters on the characteristics of the turbulent flow and the resulting cooling performance. The present study aims to analyse, through DNS simulations, the characteristics of a turbulent boundary layer in hypersonic flow with porous injection and the effects on the coolant concentration features, as well as to assess the effect of an important parameter of the porous surface, i.e. the pore size, on the cooling performance.

1.2 Film cooling and transpiration cooling techniques

Film cooling (Fitt et al., 1985) is an active cooling technique in which a coolant flow is injected into the hot hypersonic boundary layer to suppress the wall heat flux by forming a coolant film adjacent to the surface. Two injection strategies can be considered, namely injection through localised holes, i.e. effusion cooling (Baldauf et al., 2001), and through a porous structure, i.e. transpiration cooling (Langener et al., 2011). The latter is considered more efficient due to the formation of a more homogeneous coolant film via the porous structure. Injection through two-dimensional slots, as opposed to localised holes, helps reducing the three-dimensional flow effects correlated with effusion cooling, hence delaying transition (Keller et al., 2015; Keller and Kloker, 2017). Latest studies (Cerminara et al., 2020b, 2021) have found that transpiration cooling performs better then slot injection when transition to turbulence occurs, as porous injection enhances the coolant mixing within the boundary layer. However, the mechanisms through which the parameters of the porous structure and the injection patterns affect the mixing and the associated fluid dynamics features of the turbulent boundary layer have not been investigated yet. This, in turn, is crucial as it enables informed design and accurate calibration of the transpiration cooling system for optimal performance in both laminar and turbulent flows.

2. Methodology

Direct numerical simulations of the full 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 (Yee et al., 1999). The nondimensional governing equations, including the continuity equation for the coolant species, which in our case is the same as the freestream gas, i.e. air, are shown below:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_j}{\partial x_j} = 0;$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{1}{Re}\frac{\partial \tau_{ij}}{\partial x_j};$$

$$\frac{\partial \rho E}{\partial t} + \frac{\partial (\rho E + \frac{p}{\rho}) u_j}{\partial x_j} = \frac{1}{(\gamma - 1) Re Pr M^2}\frac{\partial}{\partial x_j}\frac{\partial k T}{\partial x_j} + \frac{1}{Re}\frac{\partial \tau_{ij} u_i}{\partial x_j};$$

$$\frac{\partial \rho Y_k}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho Y_k u_j - \rho D \frac{\partial Y_k}{\partial x_j}\right) = 0\ .$$

For brevity reasons, the definition and corresponding normalisation of the quantities within the system of equations is omitted here, and for all the details the reader is suggested to refer to Cerminara et al. (2020a, 2020b). Sutherland's law is used to model the viscosity. A rectangular-box computational domain for a flat plate is considered, with dimensions L_x = 300, L_y = 30, and L_z = 48. 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 x = 250 within the fully developed turbulent region, hence guaranteeing DNS resolution in all the directions, according to the work of Coleman and Sandberg (2010), being the thresholds 1, 15 and 8 for $\Delta y+$, $\Delta x+$, and $\Delta z+$, respectively. Simulations are carried out at the Reynolds number 12600, relative to the boundary-layer displacement thickness at the inlet. An isothermal wall is considered, with wall temperature of 290 K, whereas periodic conditions are imposed at the side boundaries. Details on the flow and boundary conditions are described in Cerminara et al. (2020b). Porous injection is modelled on the wall through a *x*-wise and *z*-wise periodic function, which spans the whole domain in the *z*-direction and extends from x=55 to x=100, mimicking a bed of aligned circular equally-spaced injecting pores. Two cases with different nondimensional pore diameters are considered, namely D=1.2 and D=2.4. The same blowing ratio (F=ρu_z) is considered for both cases, F =0.003. Figure 1 provides a conceptual representation of how transpiration cooling can be applied over the surface of a hypersonic vehicle (NASA X-43 is considered as an example).

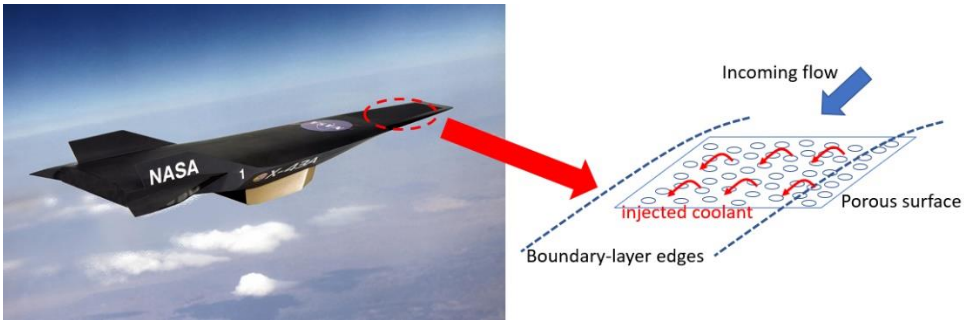

Figure 1: Representation of a bed of injecting pores in a transpiration cooling system (right) over the surface of a hypersonic vehicle (NASA X-43) (left)

3. Result analysis

Figure 2 shows instantaneous contours of the coolant concentration on the surface for both cases of pore diameter *D*=1.2 and *D*=2.4. In the region from *x*=55 to *x*=100 the bed of injecting aligned circular pores can be observed. The wedge-shaped dark-blue region downstream represents the developed turbulent wedge flow structure, in which the coolant concentration is about an order of magnitude lower than in the adjacent calmed (laminar) flow region.

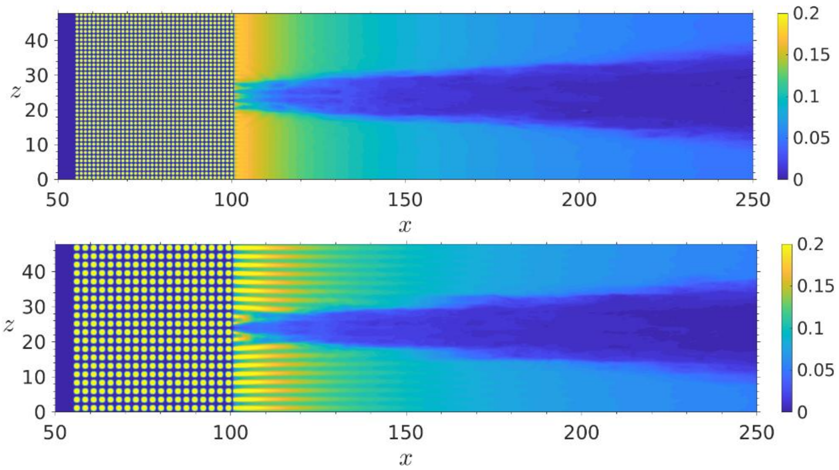

Figure 2: Instantaneous coolant concentration on the wall for *D*=1.2 (top) and *D*=2.4 (bottom)

Figure 3 shows instantaneous temperature contours within the boundary layer, which provide details of the transition patterns, including the nonlinear breakdown downstream of the porous region, and details of the turbulent structures within the turbulent wedge expanding downstream. Figure 4 shows instantaneous contours of coolant concentration at the distance *y*=0.44 from the wall, which highlights details of the coolant mixing features within the boundary layer. The coolant coming from the coolant-rich (i.e. the laminar) region appears to mix with the turbulent flow more effectively in the region just downstream of the pores (*x*>100), however the local coolant concentration gradually decays in the downstream region as the turbulent wedge spreads in the spanwise direction.

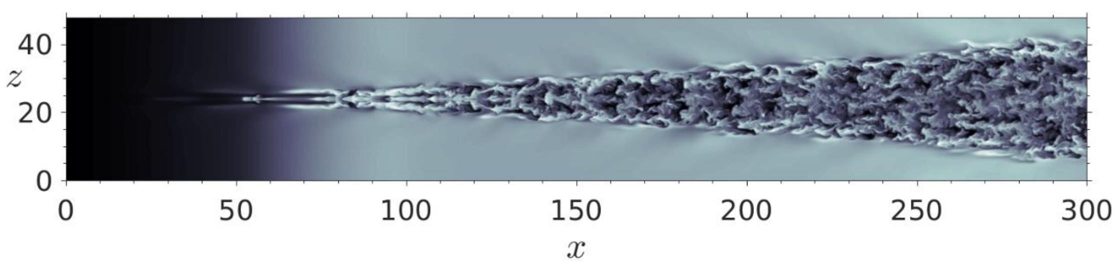

Figure 3: Instantaneous temperature field at *y*=1.4 within the boundary layer (*D*=1.2)

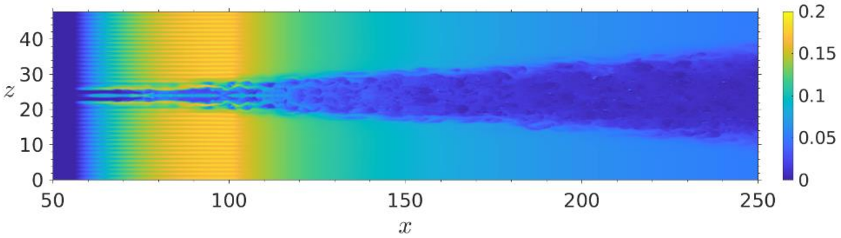

Figure 4: Instantaneous coolant concentration at *y*=0.44 within the boundary layer (*D*=1.2)

Figure 5 shows comparison of time-averaged results for the streamwise distribution of the wall heat flux and the coolant concentrations in the spanwise direction at different x stations, between the different pore size cases. As can be observed, the wall heat flux is slightly reduced from the higher pore diameter in the porous injection region, just downstream of the pores, however it reaches a similar value for both pore diameters inside the turbulent wedge. The spanwise coolant concentration profiles show that coolant concentration decays rapidly as moving from the laminar (at the sides) to the turbulent region, and that values in the laminar flow are higher for the higher pore size, whereas converge to approximately the same value in the turbulent region for both pore sizes.

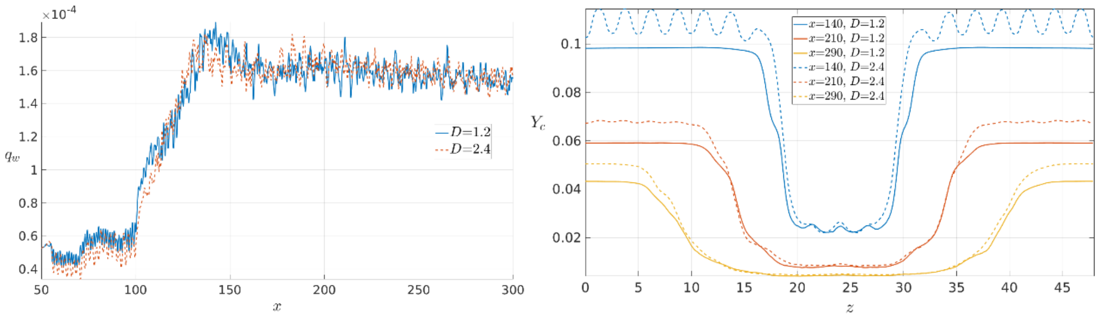

Figure 5: Time-averaged spanwise-averaged wall heat flux within the turbulent wedge (left) and time-averaged *z*-wise coolant concentrations at different *x* stations downstream of the injection region (right)

4. Discussion

Results shown in the previous section highlight the flow patterns and the influence of the pore size, including the effect on the wall heat flux. As observed in Figure 1, the coolant concentration just downstream of the pores is higher, and forms streaky structures for the higher pore diameter, *D*=2.4, whereas it appears more uniform for the lower pore size (*D*=1.2). This is consistent with the features observed in Figure 5, where pronounced oscillations with same wavelength of the pore diameter are formed in the higher pore-size case, in the laminar region, as opposed to a flatter profile shown by the smaller pore-size case. The flow rapidly transitions to turbulence due to an unsteady high-amplitude disturbance imposed on the wall upstream of the porous region. Nonlinear breakdown of the boundary layer starts above the porous region, leading to the development of the typical turbulent wedge-shaped flow structure downstream, as seen in Figure 3. Patterns of the turbulent mixing revealed in Figure 4 indicate that coolant mixing from the (laminar) sides within the turbulent wedge is more effective just downstream of the pores, where the turbulent wedge is generated. However, as moving downstream, the turbulent wedge spreads towards the sides, and mixing appears to be located only in a confined region at the edges between turbulent and laminar flow, whereas inside the turbulent wedge the level of coolant decays to very low values. This effect is due to the turbulent convective transport of coolant away from the wall, and results in both the cases with different pore size showing a similar value of the wall heat flux and coolant concentration inside the turbulent region.

5. Conclusions

DNS results have shown the flow features of a turbulent boundary layer in hypersonic flow with transpiration cooling and transition to turbulence induced from high-amplitude disturbances upstream of the porous region. It was found that a turbulent wedge rapidly develops downstream of the porous injection region, and that the effect of doubling the pore diameter produces relevant differences in the flow patterns near the porous region and within the calmed flow region, with higher observed levels of the coolant concentration. However, doubling the pore diameter provides almost negligible differences inside the turbulent wedge, where the cooling performance are strongly reduced due to the turbulent convective transport that increasingly reduces the values of coolant concentration at the wall as the wedge spreads laterally. These results indicate that an in-depth study of the flow features within the turbulent wedge, including how these affect the near-wall coolant concentration levels and wall cooling effectiveness corresponding to different parameters of the porous structure and injection patterns, is necessary in order to enable appropriate design and parameter calibration of the transpiration cooling system, which is crucial for the design and assessment of next-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

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.

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

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 (2020b): 108732.

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.

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

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.

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.

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.

Keller, M.A. and Kloker, M.J., 2017. Direct numerical simulation of foreign-gas film cooling in supersonic boundary-layer flow. AIAA Journal, 55(1), pp.99-111.

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.