**Tenth International Conference on Computational Fluid Dynamics (ICCFD10), Barcelona,Spain, July 9-13, 2018**

**ICCFD10-2018-xxxx**

# Numerical Investigation of Shock/Film-Cooling Interaction

K. Ozawa*, S. Loosen**, M. Albers**, P. S. Meysonnat**, M. Meinke**,
W. Schröder** and S. Obayashi*
Corresponding author: ozawa@edge.ifs.tohoku.ac.jp

* Institute of Fluid Science, Tohoku University, Japan.
** Institute of Aerodynamics, RWTH Aachen University, Germany.

## Introduction

In a supersonic combustion ramjet, also known as a scramjet, shock waves occur in the isolator and combustion chamber. To protect the engine's interior surfaces from intense aerodynamic heating, film-cooling can be used. The cooling effectiveness, however, is decreased by the oblique shocks interacting with the film-cooling boundary layer. Figure 1(left) sketches the basic structures of the flow field of a tangential film-cooling configuration with a shock wave [1] and Fig. 1(right) shows the $Q$ criterion with shock interaction of a simulation result. The flow field can be divided into three regions. The first region is the potential-core region just downstream of the slot, in which the maximum velocity in the cooling film is unaffected by the outer boundary layer. Further downstream, the slot boundary layer merges with the mixing layer, which forms between the film cooling flow and the outer boundary layer. In the third region, the boundary-layer region, the flow relaxes to an undisturbed turbulent boundary layer. The potential-core region is encompassed by the slot boundary layer and the mixing layer, i.e., the shear layer emanating from the lip and fed by the turbulent boundary layer. When shock waves interact with the cooling-film within the potential-core region, the shock wave changes the fundamental structure of the flow field in the vicinity of the surfaces that requires cooling, which in turn can reduce the cooling effectiveness. Therefore, this study investigates film cooling flows interacting with shock waves at different shock angles by numerical simulation and also by experiments [2].

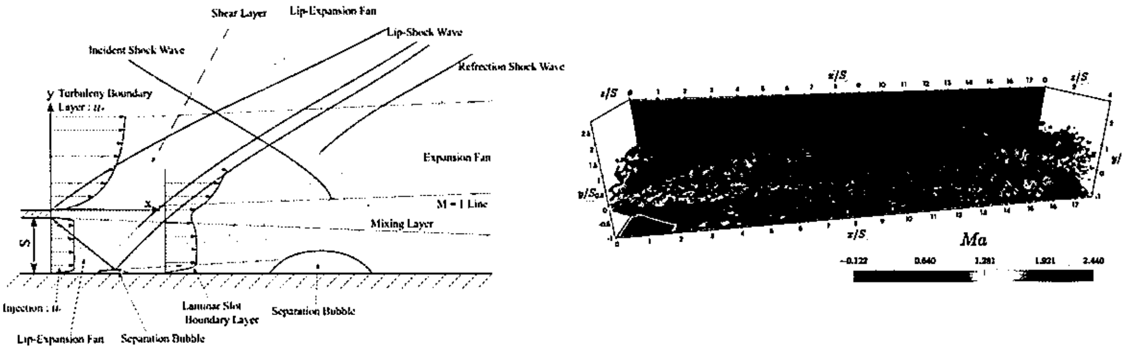

Figure 1: Flow schematic with velocity profiles indicating the shock interaction (left) and $Q$ criterion color coded with the Mach number (right).

## Geometry and Setup

Three cooling configurations with the same injection conditions are investigated by a highly resolved large-eddy simulation (LES). The cooling film is injected tangentially to the wall at an injection Mach number of $M_i$=1.8 below a supersonic turbulent boundary layer with a freestream Mach number of $M_\infty$=2.44 and a Reynolds number based on the slot height $S$ of $Re_S = (u_\infty S)/\nu_0 = 4.2 \cdot 10^4$. The total temperature ratio between the cooling flow and the mean flow is $T_{\infty,i}/T_\infty$=0.76 leading to a blowing rate of $(\rho_i u_i)/(\rho_\infty u_\infty)$=0.636. An oblique shock generated by a flow deflection of either $\beta = 5°$ or 8° resulting in a shock angle of $\sigma = 28.07°$ or 30.68° impinges upon the cooling film at a streamwise location $x/S$=17, i.e., 17 slot heights downstream of the injection. Three cases are considered in this study. A reference case without shock interaction (case I) and two cases with $\beta = 5°$ and 8° flow deflection angles (cases II and III). The simulations are performed by using an LES solver based on a finite-volume method. A mesh with approx. $300 \cdot 10^6$ mesh points is used to dicretize the domain with an equidistant spacing in the spanwise direction. A minimum wall resolution in inner coordinates of the incoming turbulent boundary layer are $\Delta x^+ = 7.47$, $\Delta y^+ = 0.99$, $\Delta z^+ = 7.56$ in the streamwise, wall-normal, and spanwise directions. The incoming turbulent boundary layer is created using the reformulated synthetic turbulent generation method [3] which generates boundary layer profiles of the mean streamwise velocity and the Reynolds shear stress in good agreement with the DNS data of [4]. The film cooling flow is ejected through a Laval nozzle such that supersonic flow conditions are obtained.

## Results

To determine the difference of the cooling effectiveness between the simulation with two shock angles, the time-resolved simulation data is analyzed and compared to preliminary experiments data. The time averaged solutions of cases I-III are compared to determine the influence of shock wave strength. The time averaged cooling effectiveness (a) and Reynolds shear stress at $x/S = 22$ (b) evaluated to measure the impact of the different shock angles on the cooling film and are shown in figure 2. More detailed results of the performed simulations of the cooling film flow will be presented at the conference.

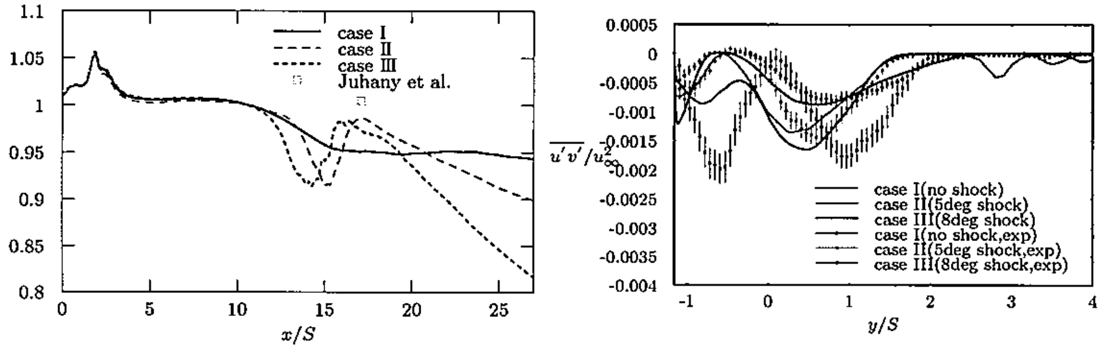

Figure 2: Cooling effectiveness $\bar{\eta}$ in the streamwise diection $x$ (left) and Reynolds shear stress component $\overline{u'v'}/u_\infty^2$ at $x/S = 22$ in wall normal direction $y/S$ (right).

## References

[1] M. Konopka, M. Meinke, and W. Schröder. Large-eddy simulation of shock/cooling-film interaction. *AIAA Journal*, 50(10):2102–2114, 2018/01/11 2012.

[2] P. Marquardt, F. Michaux, and M. Klaas. Piv measurements of supersonic slot-film cooling with shock/cooling-film interaction. 18th International Symposium on the Application of Laser and Imaging Techniques to Fluid Mechanics, LISBON, 2016.

[3] B. Roidl, M. Meinke, and W. Schröder. A reformulated synthetic turbulence generation method for a zonal RANS–LES method and its application to zero-pressure gradient boundary layers. *International Journal of Heat and Fluid Flow*, 44:28 - 40, 2013.

[4] S. Pirozzoli and M. Bernardini. Probing high-reynolds-number effects in numerical boundary layers. *Physics of Fluids*, 25(2):021704, 2013.