ExHFT-7
28 June – 03 July 2009, Krakow, Poland

DUAL INVESTIGATIONS ON THE IMPROVEMENT OF EFFUSION COOLING BY SHAPED HOLES

B. Boust[1, *], G. Lalizel[1], C. Bianchini[2], A. Ceccherini[2], G. Cinque[3], S. Colantuoni[3]
[1] Laboratoire de Combustion et de Détonique, Poitiers, France
[2] Dipartimento di Energetica "S. Stecco", Università degli Studi di Firenze, Italy
[3] Avio S.P.A., Pomigliano d'Arco, Napoli, Italy

ABSTRACT. This work is dedicated to the improvement of effusion cooling efficiency on combustion liners, by using an innovative multiperforation made of shaped holes of elliptical cross-section. Low-temperature experiments provided a spatial characterization of cooling efficiency in a suction-type wind tunnel, whereas discrete measurements were performed in the engine-like conditions of THALIE facility. A RANS computational study was also performed to give a detailed insight on the aerothermodynamics of the complex flow created by effusion holes. Experimental and numerical data confirm the enhancement of effusion cooling by shaped holes, thanks to a better coverage of the first rows of the plate, and a better individual efficiency of each hole.

Keywords: *combustor liner, effusion cooling, adiabatic efficiency, thermochromic liquid crystals*

INTRODUCTION

Due to the increasing need to reduce the emissions of aero-engines, it is necessary to optimize the design and integration of engine components. Among them, the combustion chamber focuses many of the challenges encountered. Indeed, it is the very place at which combustion and thermal phenomena pilot the emissions and efficiency of the engine. Therefore, the combustion liner faces important mechanical and thermal stresses. Moreover, the need for higher engine efficiency leads to increasing combustion temperature and pressure that emphasize stresses on the liner.
This statement points out the importance of an adapted cooling of combustion liners. For this, the use of multiperforation is a common way to create a full-coverage cooling of the liner, which is referred to as effusion cooling. However, this technique requires a significant mass flow of fresh air taken from the compressor. Thus, an adapted design is required so that the air passing through effusion holes does not affect locally the combustion rate and globally the engine efficiency.
In terms of multiperforation design, many aspects of hole geometry can be varied. The spacing, the angle, or the length to diameter ratio can be optimized by simple variation, whereas the shape of the hole itself is much harder to study due to practical difficulties in machining and controlling the shape, for instance. Former studies have pointed out experimentally the influence of the transverse dimension of the holes on cooling efficiency, either with converging slot holes called "consoles" [1] or laterally diffused holes [2]. The common feature of these design solutions is a greater extension of the hole perpendicularly to the flow, inducing a better coverage of the multiperforated wall.
This work focuses on an innovative design of multiperforation, using holes of elliptical cross-section that can be processed through laser drilling. Experiments were carried out on cooled plates to compare the efficiency of shaped to conventional holes. A RANS simulation was also carried out to give a detailed insight on the aerothermodynamics of the complex flow of the jets in crossflow.

* Corresponding author: Dr. B. Boust
Phone: + (33)549366090, Fax: + (33)549366064
E-mail address: bastien.boust@lcd.ensma.fr

EXPERIMENTAL SETUP AND MEASUREMENTS

Experiments were carried out to compare the cooling efficiency of conventional and shaped holes, using two experimental devices. The first one allows a fine spatial characterization of cooling efficiency at low temperature, whereas the second one allows a discrete measurement of cooling efficiency in engine-like conditions. Such experiments aim at quantifying the thermal behaviour of each hole geometry in terms of cooling efficiency. Practically, each hole geometry is studied using flat plates drilled with arrays of holes.

Suction-type wind tunnel

The first experiments consist of low-temperature measurements in a suction type wind tunnel.

Experimental facility. The test rig is an open-loop suction-type wind tunnel allowing the measurement of heat transfer, cooling efficiency and pressure loss on multiperforated plates, by controlling completely the mainstream flow and the coolant flow.
The mainstream air flow, at atmospheric pressure and ambient temperature, passes first through a 24 kW controlled electric heater, where desired temperature is reached and kept constant, before entering the PolyMethyl-MethAcrylate test section depicted in Figure 1, through a setting valve.
The coolant flow temperature is controlled by mixing heated air and cool air. Four rotary vane vacuum pumps (total power installed 59 kW) provide the suction for a maximum mass flow rate of 0.50 kg/s. Both flow rates are set up by guiding the motor speeds between 300 and 1200 rpm, and throttling the remote controlled motorized valves. The air temperature exiting from the heater is controlled by means of a four wire resistance temperature detector (RTD Pt-100).
The rig allows performing measurements on test samples made of adiabatic (stereolithography resin, STL) and conductive material (stainless steel, AISI 321), see Table 1.

Table 1
Geometric features of the multiperforated plates

Multiperforated plate	with circular holes	with shaped holes
Plate thickness (mm)	12.7	12.7
Number of holes	38	28
Number of rows	8	5
Streamwise to spanwise pitch ratio	2.3	2.0
Porosity	0.036	0.0375
Angle of the holes (°)	17.0	17.0
Hydraulic diameter of the holes (mm)	2.65	3.385
Length to diameter ratio L/D	16.39	12.8

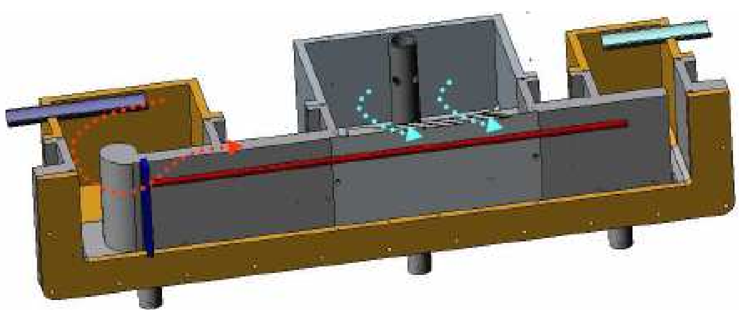


Figure 1. Suction-type wind tunnel

Diagnostics. Tests are run once flow rates, pressures and temperatures have reached steady conditions. Adiabatic and overall cooling efficiencies, denoted resp. as η_{aw} and η_{ov} (see equations (1) and (2)), are evaluated resp. on adiabatic and conductive plate using thermochromic liquid crystals that provide surface temperature T_w. Efficiencies are computed starting from the measured wall temperature values by means of different post-processing procedures accurately described by [3]. The final result of such procedures is the achievement of η_{aw} values once the plate heat loss due to holes heat sink effect is evaluated.

$$\eta_{aw} = \frac{T_{main} - T_{aw}}{T_{main} - T_c} \tag{1}$$

$$\eta_{ov} = \frac{T_{main} - T_w}{T_{main} - T_c} \tag{2}$$

Conditions investigated. All tests have been carried out setting approximately the coolant temperature at 306 K and the mainstream temperature at 317 K and 322 K, resp. for adiabatic and conductive tests. Experimental survey has been performed setting jet Reynolds number at 12500, and imposing three values of blowing ratio close to engine operating conditions (BR = 5–7–9). Blowing ratio is commonly defined from flow momentum in the holes and in the mainstream.

$$BR = \frac{(\rho V)_{hole}}{(\rho V)_{main}} \tag{3}$$

High-temperature wind tunnel THALIE

In the second set of experiments, conductive plates are exposed to realistic engine conditions.

Experimental facility. Experiments are carried out using THALIE, that is a high-temperature wind tunnel designed to study THermal and Aerodynamic features of Experimental Liners, see Figure 2. THALIE generates experimental conditions close to those encountered in the vicinity of combustor liners, in terms of gas velocity, temperature, pressure, and pressure drop across the liner. The test section is equipped with apertures, allowing optical measurements (PIV, LDV, visualization, fluorescence, thermography), as well as intrusive measurements (gas sampling, thermocouples).

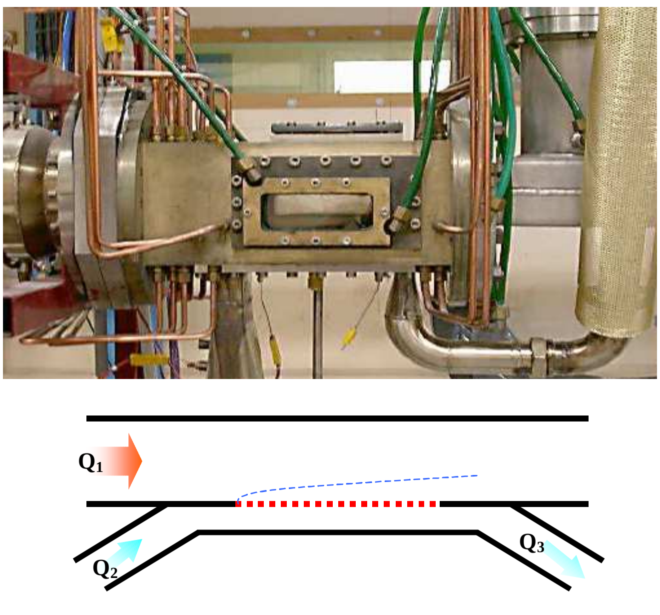


Figure 2. High-temperature wind tunnel THALIE

During an experiment, a multiperforated plate is submitted to a hot gas flow and a coolant flow. The hot flow, Q_1, is made of combustion products supplied by a tubular combustor fuelled with kerosene. The cooling air flow, Q_2, can be heated electrically. The temperatures of both the hot and cooling flows are P.I.D. controlled, ensuring the stability and reliability of experiments. The flow passing through the multiperforated plate, Q_h, is only a fraction of the cooling flow. It is driven by controlling the pressure drop across the plate, ΔP. Q_h is evaluated as the difference between the incoming (Q_2) and outgoing (Q_3) coolant flows. The controlled mass flow rates (Q_1, Q_2 and Q_3) are measured using vortex-type flowmeters. T_1 and T_2 are measured by K-type thermocouples.
Experiments are carried out twice, i.e. once using a plate with conventional holes, and once using a plate with shaped holes. The multiperforated plates are manufactured by AVIO S.p.A. with thermal coating and laser drilling technology (see Table 2). Overall, both plates have similar properties, and similitude rules have been applied to scale the multiperforation. The experimental conditions summarized in Table 3 provide a wide range of blowing ratio, BR = 1–10.

Table 2
Geometric features of the multiperforated plates

Multiperforated plate	with circular holes	with shaped holes
Plate thickness (mm)	2.5	2.5
Number of holes	569	234
Number of rows	17	12
Streamwise to spanwise pitch ratio	2.1	2.0
Total cross-section of the holes (mm²)	149	149
Angle of the holes (°)	18.5	17.3
Hydraulic diameter of the holes (mm)	0.58	0.76
Length to diameter ratio	13.6	11.1

Table 3
Operating conditions of the experiments

	Hot flow	Cooling flow	Hole flow
Subscript	1	2	h
Reynolds number	60,000 – 80,000	10,000 – 100,000	0 – 20,000
Cross-section (mm)	126×72	100×20	
Mass flow rate (kg/s)	0.4	0 – 0.2	
Temperature (K)	1000	288	
Pressure (bar)	3.0	3.0 – 3.5	

Diagnostics. The cooling efficiency of the multiperforated plate is evaluated by means of adiabatic and overall efficiency. Overall efficiency, η_{ov}, takes into account convection as well as conduction and radiation. Evaluating η_{ov} requires the measurement of wall temperature, T_w, see equation (4). Thus, the multiperforated zone is instrumented with K-type thermocouples of diameter 0.5 mm, sheathed with stainless steel. They provide the wall temperature on the cold side of the plate.

$$\eta_{ov} = \frac{T_1 - T_w}{T_1 - T_2} \tag{4}$$

Adiabatic efficiency, η_{ad}, is the cooling efficiency of an equivalent adiabatic plate. It is evaluated by assuming the analogy between heat and mass transfer, which consists of equal molecular and thermal diffusivities, i.e. a unity Lewis number. Measuring adiabatic efficiency requires the local concentration of a reference gaseous species at the wall surface on the hot side, C_w, as well as inside the primary and secondary flows, respectively C_1 and C_2, see equation (5).

$$\eta_{ad} = \frac{C_1 - C_w}{C_1 - C_2} \tag{5}$$

For this, a water-cooled sampling probe of diameter 6.0 mm is set inside the hot flow by the top aperture and fixed to a displacement system driven by a computer. The sampling orifice of diameter 0.7 mm is located at the conical tip of the probe. The reference gaseous species selected for sampling is CO_2, which concentration is constant in the hot flow and null in the cool flow, i.e. C_2=0. CO_2 is probed by an inline gas analyser (COSMA) based on the absorption of infrared radiation.

NUMERICAL SIMULATION

This numerical study was performed using a solver developed into the open-source environment for continuum mechanics analysis named OpenFOAM® [4]. The toolkit implements operator-based implicit and explicit 2nd-order and 4th-order finite volume discretization in 3-dimensional space.
These simulations were run using the steady-state compressible SIMPLE-like solver developed at University of Florence and validated in [5]. The energy equation is solved in terms of total-enthalpy in order to avoid energy losses in high acceleration regions namely the coolant flow into the ducts. The two-layer turbulence model by Rodi was used to model turbulent behavior with Norris and Reynolds closure formulas [6]. Even though the gain compared to standard two-equation eddy viscosity models is considerable in terms of well reproducing near-wall behavior, this model, further referred to as isotropic, suffers the well-known deficiency of the eddy viscosity models to correctly predict the lateral spreading of the jet in crossflow. The same model was thus tested implementing an algebraic correction to increase the lateral diffusion of momentum, energy and turbulence as proposed by [7]. This corrected version is later referred to as the anisotropic model.
For conductive tests, the coupling between solid and fluid was achieved in an explicit manner: at each iteration step a new value of total temperature on the boundary is calculated imposing the continuity of the thermal flux across the interface. A weighted average based on face overlapping area allows the use of non-conformal interfaces between fluid and solid domain. More details on the coupling procedure and the treatment of generic grid interfaces can be found in [8].
The meshes used to perform such calculations finely discretize boundary layer reaching y^+ values up to 1 for all walls of interest, namely the two sides of the plate and the hole ducts. In order to save computational cost all other walls were treated as inviscid wall (symmetry planes) thus avoiding the clustering. Moreover two symmetry planes were inserted at the center of holes, and only one spanwise pitch was simulated in order to save computational cost and to better impose stationary behavior, see Figure 3. The total grid size is around 3 million cells for both the conventional and the shaped geometries. Apart from the different dimensions, the numerical setup to reproduce the two experimental rigs only differs in the boundary conditions for the abduction of coolant. For the suction-type tunnel, constant pressure plenum is simulated imposing no cross-flow and a large low-speed inlet area, see Figure 3. As for THALIE, the same conditions as for the mainstream are applied to the coolant. The boundary conditions were extracted from the experimental ones. For the low-speed wind tunnel, the blowing ratio could be imposed and the pressure drop between plenum and mainstream taken as results. For the THALIE rig, the procedure was inverse, thus it was not possible to investigate numerically the two rig configurations exactly at the same blowing ratio.

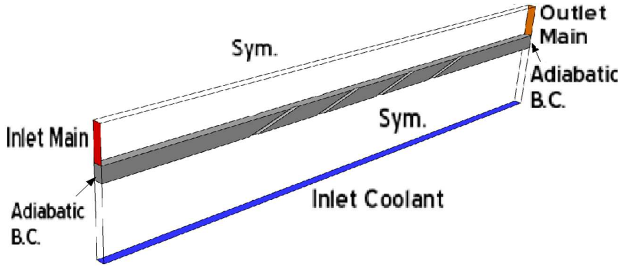


Figure 3. Overview of boundary conditions

RESULTS AND DISCUSSION

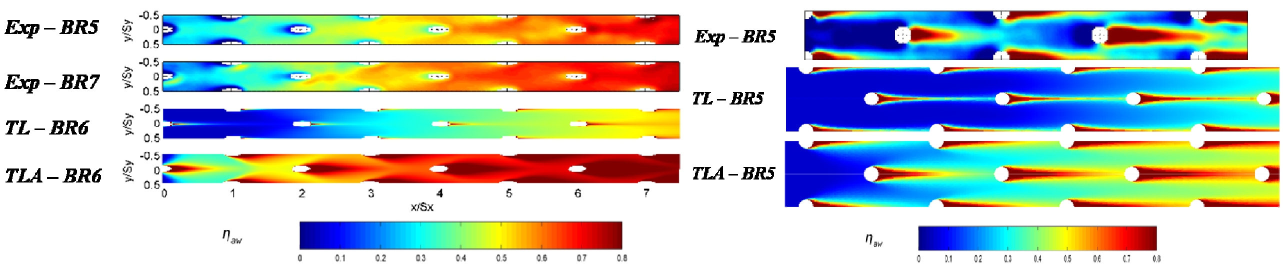
In this section, the results of experiments reported along with the results of numerical simulation are expected to determine whether shaped holes actually improve the efficiency of effusion cooling.

Suction-type wind tunnel

On such rig, the fine resolution of temperature measurements allows investigating the spatial distribution of efficiency. Results are reported in terms of spanwise averaged efficiency. However, efficiency is so uniform for conductive tests that only spanwise averaged profiles are reported.

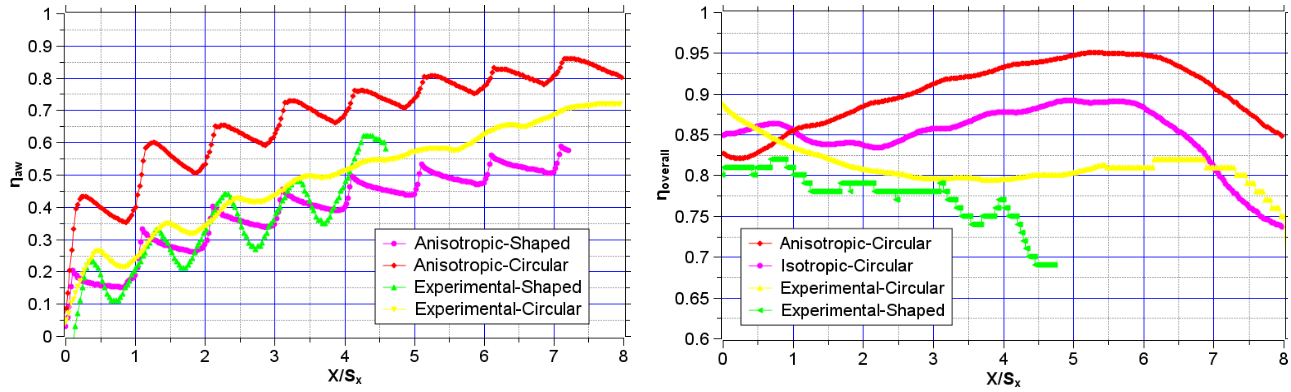
Adiabatic efficiency maps. In conventional geometry, experimental results at BR=5–7 are reported together with numerical simulations at BR=6, see Figure 4a. Firstly, experimental maps (Exp) exhibit a cold zone upstream of the holes due to the effect of the coolant ducts inside the plate, meaning that the hypothesis of adiabatic behaviour is not completely respected. Secondly, both turbulence models fail in predicting the downstream film development. While the isotropic model (TL) under-predicts the lateral diffusion of jets, the anisotropic correction (TLA) emphasizes this effect too much. In shaped configuration, the shape of coolant wake is better simulated by the anisotropic model, see Figure 4b. By comparing both geometries, the spanwise profile of conventional geometry is obviously more uniform. Shaped geometry should therefore be studied carefully as far as thermal stresses are concerned.

Spanwise averaged efficiency. The profiles of adiabatic and overall efficiency at BR=5 are shown in Figure 5. The anisotropic model simulates well the efficiency of shaped geometry, but overestimates that of conventional geometry. This result is confuting the findings on conventional geometry for higher angles and lower blowing ratios [9]. As for overall efficiency, the conventional plate is better cooled due to a higher heat sink effect connected with a higher L/D. As can be seen comparing experiments with computations, the different modeling of solid plate far form the drilled part modifies strongly the profile of overall efficiency. Although conventional geometry behaves better in terms of cooling performance, this is only related to the specific test conditions. A deeper survey points out that shaped configuration is better performing at higher blowing ratios [10].



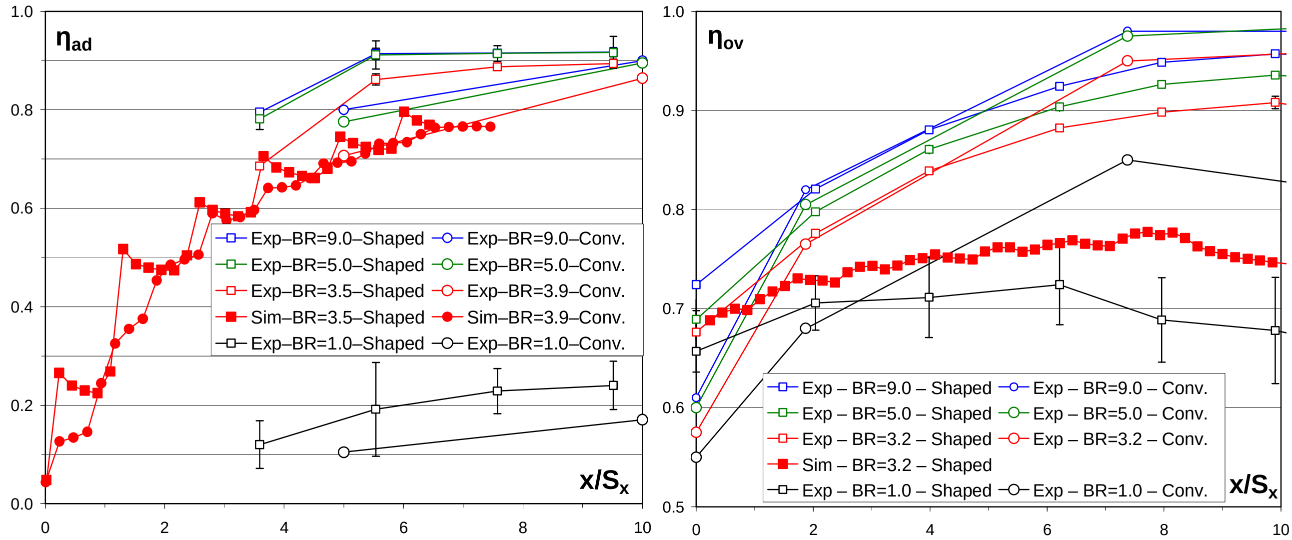
a) b)

Figure 4. Adiabatic efficiency maps – a) Conventional case, b) Shaped case



a) b)

Figure 5. Spanwise averaged efficiency at BR=5 – a) Adiabatic, b) Overall



a) b)

Figure 6. Streamwise profile of cooling efficiency – a) Adiabatic, b) Overall

High temperature wind tunnel THALIE

Measurements of adiabatic and overall cooling efficiency allow the comparison between the two multiperforation geometries, i.e. with conventional or shaped holes.

Adiabatic efficiency quantifies the ability of the plate to perform effusion cooling through the holes, regardless of conduction and radiation phenomena. The streamwise profile of adiabatic efficiency along the plate is presented for experimental and numerical data (see Figure 6a). As usual, adiabatic efficiency increases with increasing blowing ratio. Experimental data show a high uncertainty at low BR, but converge quickly towards an asymptotic curve near BR=5. The experimental and numerical profiles of adiabatic efficiency indicate that cooling is enhanced by shaped holes. The benefit in efficiency decreases along the plate, from 0.08 at 5th row to 0.02 at 10th row. Especially, this benefit exceeds 0.15 at the first two rows due to a strong increase in efficiency in the wake of shaped holes.

The evolution of overall efficiency versus BR is similar to that of adiabatic efficiency, because increasing BR improves effusion cooling (see Figure 6b). Experimental curves confirm that shaped holes enhance cooling in the first rows. Cooling efficiency is similar for both geometries between 3rd and 6th rows. Finally, conventional holes have better cooling performance beyond 7th row. The difference between simulation and experiments may be due to uncertainties in determining low values of BR in 1–3, whereas overall efficiency increases strongly with BR. Moreover, the higher L/D of conventional holes enhances the heat sink effect in conventional geometry due to a greater inner surface of the conventional holes.

CONCLUSIONS

Experiments and simulations have been carried out on two geometries of effusion cooling, involving conventional holes and holes of elliptical cross-section. Two configurations have been addressed, providing high blowing ratios in either low-temperature or realistic engine conditions.
This survey points out many features of effusion cooling by shaped holes. Experiments and simulation show that they have a better individual cooling behaviour inducing higher efficiency in the first rows (up to 15%), but conventional holes lead to a better film formation that provides superior thermal protection after several rows of holes. Additionally, this improvement in cooling efficiency is proved to depend strongly on physical conditions, such as blowing ratio and length to diameter ratio [10].
Finally, shaped holes of elliptical cross-section provide a promising way for improving effusion cooling. Indeed, this design recalls not only the shape but also the high thermal efficiency of "consoles" [1]. Further studies should check that such cooling improvement is obtained without penalty to the mechanical strength requirements induced by thermal stresses of combustion liners.

ACKNOWLEDGEMENTS

The present work was supported by the European Commission as part of FP6 STREP INTELLECT D.M., which is gratefully acknowledged together with consortium partners.

REFERENCES

1. Sargison, J. E., Guo, S. M., Oldfield, M. L. G., Lock, G. D., Rawlinson, A. J., A converging slot-hole film cooling geometry – Part 1: Low-speed flat plat heat transfer and loss, *ASME J. Turbomachinery*, Vol. 124, No. 3, pp 453-460, 2002.
2. Hyams, D. G., Leylek, J. H., A detailed analysis of film cooling physics: Part III – Streamwise injection with shaped holes, *ASME J. Turbomachinery*, Vol. 122, No. 1, pp 122-132, 2000.
3. Facchini, B., Tarchi, L., Toni, L., A post processing procedure for the evaluation of adiabatic and overall effectiveness of effusion cooling geometries, *Proceedings of the 19th Symposium on measurement techniques*, Rhode S. Genèse, 2008.
4. Jasak, H., Weller, H. G., Nordin, N., In-cylinder CFD simulation using a c++ object-oriented toolkit, *SAE techincal paper* 2004-01-0110, 2004.
5. Mangani, L., Bianchini, C., Andreini, A., Facchini, B., Development and validation of a c++ object oriented CFD code for heat transfer analysis, *ASME-JSME Summer Heat Transfer Conference, Vancouver,* 2007.
6. Rodi, W., Experience with two-layer models combining k-epsilon model with a one-equation model near the wall, *29th Aerospace Sciences Meeting*, AIAA 91-0216, 1991.
7. Lakehal, D., Theodoris, G. S., Rodi, W., Three-dimensional flow and heat transfer calculations of film cooling at the leading edge of a symmetrical turbine blade model, *International journal of heat and fluid flow*, Vol. 22, No. 5, pp 113-122, 2001.
8. Bianchini, C., Facchini, B., Mangani, L., Conjugate heat transfer analysis of an internally cooled turbine blade with an object oriented CFD code, *European Turbomachinery Congress*, Graz, 2009.
9. Andreini, A., Mangani, L., Application of an object-oriented CFD code to heat transfer analysis, *ASME paper* GT 2008-51118, 2008.
10. Facchini, B., Tarchi, L., Toni, L., Cinque, G., Colantuoni, S., Investigation of circular and shaped effusion cooling arrays for combustor liner application – Part 1: Experimental analysis, *ASME paper* GT 2009-60037, 2009.