Direct and Large-Eddy Simulations of a Turbulent Flow with Effusion

S. Mendez[1], F. Nicoud[1] and P. Miron[2]

[1] CERFACS, 42, avenue Gaspard Coriolis. 31057 Toulouse cedex 1. France. `mendez@cerfacs.fr`
[2] TURBOMECA, 64511 Bordes Cedex, France `mironpetre@yahoo.fr`

Abstract

LES calculations results are reported of the flow created by an infinite effusion plate, with staggered holes inclined at an angle of 30 deg to the main flow. Two original methods are proposed to generate the flow in a periodic configuration. Results for mean velocity and velocity fluctuations are compared with measurements made on a large-scale isothermal experiment.

1 Introduction

In almost all the systems where combustion occurs, solid boundaries need to be cooled. One possibility often chosen in gas turbines is to use multiperforated walls to produce the necessary cooling. In this approach, fresh air coming from the casing goes through the perforations and enters the combustion chamber [5]. The associated micro-jets coalesce to give a film that protects the internal wall face from the hot gases. The number of submillimetric holes is far too large to allow a complete description of the generation/coalescence of the jets when computing the 3D turbulent reacting simulation within the burner. Effusion is however known to have drastic effects on the whole flow structure. As a consequence, new wall functions for turbulent flows with effusion are required to perform predictive full scale computations.

One major difficulty in developing wall functions is that the boundary fluxes depend on the details of the turbulent flow structure between the solid boundary and the fully turbulent zone. Unfortunately, the configuration of full-coverage film cooling is difficult to treat:

- the perforation imposes small scales structures that are out of reach of current experimental devices and the thermal conditions in actual gas turbines make any measurement very challenging.
- the number of holes make accurate simulations difficult to perform.

This explains the lack of data concerning full-coverage film cooling through discrete holes. Most available aerodynamic measurements deal with large-scale isothermal flows [5, 7, 13]. On the contrary, experimental studies about the thermal behaviour (evaluation of cooling effectiveness and heat transfer coefficient) do not provide flow measurements [3], or too coarse ones [12].

RANS (Reynolds Average Navier-Stokes) simulations of full-coverage film cooling with a several rows (10 rows for Harrington *et al.* [6]) have been performed. However, as noted by Acharya *et al.* [2], the issues of modeling errors and the non-universality of the turbulence models do not allow to consider RANS codes as predictive tools. Large-Eddy Simulations (LES) or Direct Numerical Simulations (DNS) only treated the configuration of a single jet in crossflow. Moreover, numerical simulations often study the case of large hole length-to-diameter ratios, while effusion cooling for combustion chamber walls is done through short holes, because of the small thickness of the plates in aeronautical applications. This makes necessary to compute both the aspiration and the injection sides of the perforated plate, in order to avoid unrelevant assumptions on the flow in the hole.

In order to gain precise information about the behaviour of the flow near a perforated plate, Large-Eddy Simulations are performed: DNS or Wall-resolved LES can be used to generate precise and detailed data of generic turbulent flows under realistic operating conditions, with no limitations due to the size of the configuration or to difficulties to realise measurements in a hot flow. In this paper, two computational methodologies are proposed to perform calculations of an effusion flow and results are compared with the TURBOMECA experimental database, generated on the LARA test rig [7]. Present results only concern the injection side of the flow.

2 Computational Methodology

For effusion cooling studies, experimental test rigs are generally divided into two channels: one represents the combustion chamber, with a primary flow of hot gases and the other represents the casing, with a secondary flow of cooling air. The plate between the channels is perforated. Because the pressure is higher in the casing side, cooling air is injected through the perforated plate.

Results in the open literature show that the effusion flow highly depends on the configuration of interest: the flow generated by a ten-row plate would be different from the one generated by a twenty-row plate. This situation is hardly tractable from a modeling point of view and we decided to simplify the problem by considering the asymptotic case where the number of rows is infinitely large. The present simulation is then designed to reproduce this asymptotic case of a turbulent flow with effusion around an infinite plate. This choice presents several advantages: the infinite plate can be reduced to a domain containing only one perforation, with periodic boundary conditions to reproduce the whole geometry of an infinite plate, as it is suggested in

Fig. 1 and the difficult question of the inlet and outlet boundary conditions in turbulent simulations (see [9]) is avoided.

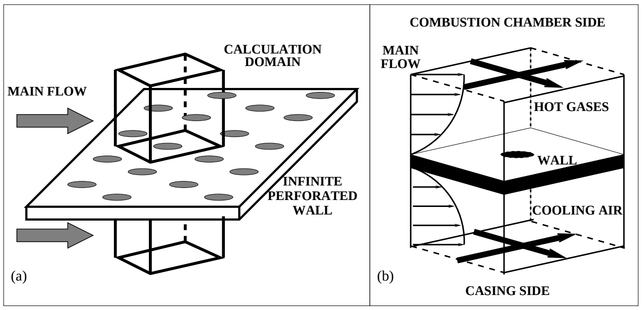

Fig. 1. From the infinite plate to the "bi-periodic" calculation domain. (a) Geometry of the infinite perforated wall. (b) Calculation domain centred on a perforation; the bold arrows correspond to the periodic directions.

With such a periodic calculation domain, the objective is to have information about the structure of the flow far from the first rows, when the film is established. However, this periodic option raises a problem: natural mechanisms that drive the flow, such as pressure gradients, are absent. The flow has to be generated artificially. The purpose of next sections is to describe two treatments used to drive the flow.

2.1 Main flows

For a classical channel flow, a volumetric source term is added to the momentum conservation equation in order to mimic the effect of the mean streamwise pressure gradient that would exist in a non-periodic configuration. This is a very classical method for channel or pipe flow simulations. The source term is usually constant over space. For example, it can have the following form:

$$S_{(\rho U)} = \frac{(\rho U_{target} - \rho U_{mean})}{\tau} \tag{1}$$

The source term compares a target for momentum, ρU_{target}, with the average momentum in the channel, ρU_{mean}. A relaxation time τ characterizes the rapidity with which ρU_{mean} tends towards its target value. Naturally, this treatment is done for one channel. This approach can be generalized in the case of an effusion configuration, having a source term of the previous form for each channel: therefore, the source term on momentum is constant by part, with two distinct values for the cold and the hot sides. No source term is applied in the hole.

2.2 Injection

In experiments, channels are bounded by impermeable walls at the top and at the bottom. If used in conjunction with periodic boundary conditions in the tangential directions, this outer condition prevents the flow from reaching a statistically steady state with effusion, because the net mass flux through the perforation tends to eliminate the pressure drop between the cold and the hot domains. In order to sustain the secondary effusion flow in periodic LES, two different strategies are investigated:

First option CST method: Constant Source Terms

In this method, boundary conditions at the top and at the bottom are walls. Exactly as it is done for momentum, constant (over each half of the domain) source terms on pressure and density are used in order to drive these quantities towards prescribed target values consistent with the operating point that is being studied. As temperature, pressure and density are linked together via the state law, controlling the couple (P,ρ) allows to control the couple (P,T). They have the same form as for the momentum source term.

Second option BC method: Boundary Conditions

This method is based on the use of boundary conditions. In this method, the physical boundaries that bound the experiment are replaced by characteristic-based [11] freestream boundary conditions. These boundary conditions allow to define pressure, density and velocity at the top and the bottom boundaries in order to impose the appropriate mean vertical flow rate.

3 Details of Numerical Simulations

The computational domain is designed as the smallest domain that can reproduce the geometry of an infinite plate with staggered perforations, as it is shown in Fig. 1. All simulations are carried out with the AVBP code [1]. It is a fully explicit cell-vertex type code that solves the compressible Navier-Stokes equations on unstructured meshes for the conservative variables (mass density, momentum and total energy). AVBP is dedicated to LES and DNS, and it has been widely used and validated in the past years in all kinds of configurations. The present simulations are based on the WALE sub-grid scale model [10]. A coarse grid (grid 1) is first used to evaluate the ability of the two methods (CST and BC) to generate an effusion flow. It contains 150,000 tetrahedral cells. Finer computations made on a grid containing 1,500,000 tetrahedral cells (grid 2) are then performed. In this second grid, fifteen points describe the diameter of the hole and on average the first off-wall point is situated at $y^+ \approx 5$. The numerical scheme is the Lax-Wendroff scheme (second order in time and space) for the calculations on grid 1 and the TTGC scheme [4] (third order in time and space) for grid 2: this scheme was specifically developed to handle unsteady turbulent flows with unstructured meshes.

4 Results and Discussion

4.1 Operating point

Both methods are tested and results are compared with experimental measurements obtained by Laser Doppler Anemometry. The configuration that is considered in this paper corresponds to the geometry studied by Miron [7]. The study focuses on the case of a large-scale isothermal plate, with a hole diameter of d=5 mm (0.5 mm is a common value for gas turbines applications). Holes are spaced 6.74 diameters apart in the spanwise direction and 5.84 diameters apart in the streamwise direction. The thickness of the plate is 10 mm and holes are angled at 30 deg with the plate: they are short holes, with a length-to-diameter ratio of 1.73.

This is an isothermal experiment: both the primary flow, which represents the burned gases flow inside the combustion chamber, and the secondary flow, which represents the cold air coming from the compressor, are at the same temperature. The main aerodynamical parameters, given for the region upstream the perforations, are summarized here:

- The Reynolds number for the primary flow (based on the duct centerline velocity and the half height of the rectangular duct) is Re=17750.
- The Reynolds number for the secondary flow (based on the duct centerline velocity and the half height of the rectangular duct) is Re=8900.
- The pressure drop across the plate is 42 Pa.

In simulations, these parameters are fixed. The behavior of the flow in the hole results from the calculations. Numerical results are compared with measurements performed at the ninth row of the experiment. Further details about this experiment can be found in [7] and [8].

4.2 Comparisons between experimental and numerical results

Measurements on the LARA test rig show that effusion jets interact to form a film that protects the plate from the primary flow (hot gases in real cases). Only the structure of the film far from the first rows is interesting for comparisons with numerical simulations. After a few rows, downstream the hole, mean streamwise velocity profiles are characterized by two peaks: the first one, next to the wall, represents the jet core. It is the result of the interaction between the jet coming out of the hole and the film. The second peak represents the film core, which is the result of the interaction of all former jets with the main flow.

Figure 2 shows the mean velocity profiles at 3 diameters downstream the hole center, on the centerline plane. Symbols correspond to measurements, continuous lines to LES with the BC method and dotted lines to LES with the CST method, using grid 1. Both methods show their ability to reproduce the general topology of the flow, even on a coarse grid . Even if they are

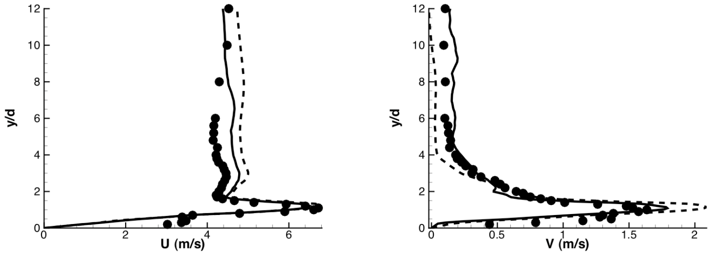

Fig. 2. Mean streamwise velocity profiles (U) and mean vertical velocity profiles (V) 3d downstream the hole, on the centerline plane.

very different, the two methods give close results. Furthermore, measurements and results of the computations are very similar. They show the same global behaviour and even the same orders for the mean velocity: the strength and the penetration of the jets are well reproduced, even if results with the BC method show a better agreement with experimental measurements.

Finer computations were performed with the BC method. Improvements are observed concerning the physics of the flow (not shown): the velocity field in the jet shows a realistic form: separation of the jet due to the sharp-edged, inclined inlet along the downstream portion of the hole is reproduced. At the outlet of the hole, the jet lift-off and the entrainment process [13] are observed. Figure 3 shows the mean and the root mean square velocity profiles in the two directions available in the LARA database (streamwise and vertical). Profiles are shown at 3 diameters downstream the jet, on the centerline plane (symbols correspond to measurements, continuous lines to LES with the BC method, using grid 2).

Figure 3.a shows two different trends for the mean streamwise velocity U: first, the behavior in the near-wall region is quite well reproduced and the velocity peak due to the jet is situated as in the experiment. This ability to describe the near-wall region is crucial because the aim is to get information about what happens in this region, in which measurements are very difficult to perform. Larger differences can be found above the jet, in the film core region. It is believed that it is mainly due to the difference between the cases of interest. Simulations characterize the flow around an infinite perforated plate, while measurements are made at the ninth row of the test rig. This effect of accumulation for mean streamwise velocity is coherent with what is observed experimentally [7]. The number of rows has an effect on the mean velocity profile above the peak due to the jet: the velocity of the film core tends to increase with the number of rows. For quantities that are not directly affected by the effect of accumulation due to the infinite configuration, comparisons between experiments and simulations show good agreement, as shown for the mean vertical velocity or for the root mean square velocities.

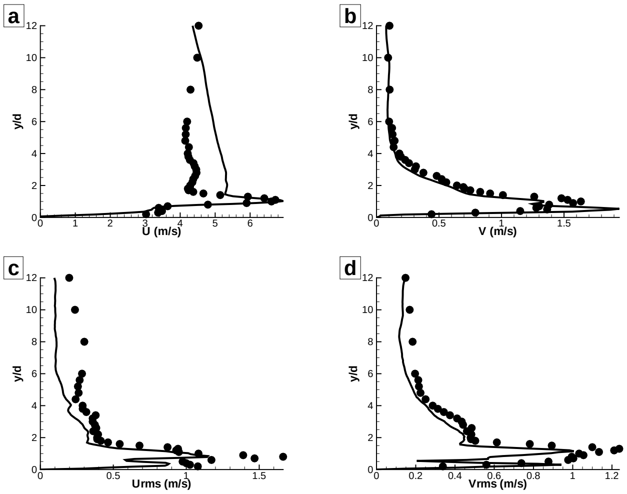

Fig. 3. Numerical (BC method on grid 2) and experimental profiles, 3d downstream the hole, on the centerline plane. a). Mean streamwise velocity U. b). Mean vertical velocity V. c). RMS streamwise velocity Urms. d). RMS vertical velocity Vrms.

5 Conclusion

To compute realistic cases of effusion cooling in the generic configuration of an infinite plate, two original methods are proposed. Although the methods are very different, their results are similar. Computational velocity fields are compared with measurements coming from the TURBOMECA experimental database generated by LDA on the LARA experiment. This configuration is an isothermal effusion flow through large-scale holes. Coarse computations reproduce the general topology of the flow.

A second finer grid is used to perform simulations with the BC method. The use of the second grid leads to improvements on the physical structure of the flow and mean velocity levels are modified and show much more details about the flow. Velocity gradients at the wall are correctly predicted. Mean streamwise velocity shows an effect of the infinite plate, with differences on the film core prediction, compared to the experimental data. RMS velocity levels show a good general behaviour, with good prediction of the trends observed in the experiment.

Future work consists in performing simulations in an anisothermal configuration, that are necessary to get information about the thermal behavior of the flow around the perforated plate. The objective is also to carry out Direct Numerical Simulations, in order to avoid the question of the sub-grid model needed for LES computations. Computational results will then be used to derive a wall function for effusion walls.

6 Acknowledgements

The authors are grateful to the European Community for funding this work under the project INTELLECT-DM (EU Project AST3-CT-2003-502961), and to the Centre Informatique National pour l'Enseignement Supérieur (CINES) for the access to supercomputer facilities.

References

1. *AVBP http://www.cerfacs.fr/cfd/avbp_code.php.*
2. S. Acharya, M. Tyagi, and A. Hoda. Flow and heat transfer predictions for film cooling. *HEAT TRANSFER IN GAS TURBINE SYSTEMS. Annals of the New York Academy of Sciences*, 934:110–125, 2001.
3. F. Bazdidi-Tehrani and G.E. Andrews. Full-coverage discrete hole film cooling : investigation of the effect of variable density ratio. *Journal of Engineering for Gas Turbines and Power*, 116:587–596, 1994.
4. O. Colin and M. Rudgyard. Development of high-order Taylor-Galerkin schemes for unsteady calculations. *J. Comp. Physics*, 162(2):338–371, 2000.
5. R.J. Goldstein. *Advances in Heat Transfer.* Academic Press, New-York and London, 1971.
6. M. K Harrington, M. A. McWaters, D. G. Bogard, Lemmon C. A., and K. A. Thole. Full-coverage film cooling with short normal injection holes. *ASME TURBOEXPO 2001. 2001-GT-0130*, 2001.
7. P. Miron. *Étude expérimentale des lois de parois et du film de refroidissement produit par une zone multiperforée sur une paroi plane.* PhD thesis, Université de Pau et des Pays de l'Adour, 2005.
8. P. Miron, C. Berat, and V. Sabelnikov. Effect of blowing rate on the film cooling coverage on a multi-holed plate: application on combustor walls. In *Eighth International Conference on Heat Transfer. Lisbon, Portugal*, 2004.
9. P. Moin and K. Mahesh. Direct numerical simulation: A tool in turbulence research. *Annu. Rev Fluid Mech.*, 30(539-578), 1998.
10. F. Nicoud and F. Ducros. Subgrid-scale stress modelling based on the square of the velocity gradient tensor. *Flow, Turbulence and Combustion*, 62(3):183–200, 1999.
11. T. Poinsot and S. Lele. Boundary conditions for direct simulations of compressible viscous flows. *J. Comp. Physics*, vol.101(1):104–129, 1992.
12. S. Rouvreau. *Étude expérimentale de la structure moyenne et instantanée d'un film produit par une zone multiperforée sur une paroi plane. Application au refroidissement des chambres de combustion des moteurs aeronautiques.* PhD thesis, E.N.S.M.A. et Faculté des Sciences Fondamentales et Appliquées, 2001.
13. S. Yavuzkurt, R.J. Moffat, and W.M. Kays. Full coverage film cooling. Part 1. Three-dimensional measurements of turbulence structure. *J. Fluid Mech*, 101:129–158, 1980.