International Journal of Thermal Sciences 118 (2017) 1–11

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

journal homepage: www.elsevier.com/locate/ijts

Confined impinging air jet on a heated cylinder at low Mach number

E. Schall [a,*], N. Chauchat [a], M. Mory [b]

[a] Univ Pau & Pays Adour, laboratoire des Sciences de l'Ingénieur Appliquées à la Mécanique et au génie Electrique – IPRA, EA4581, 64000, Pau, France
[b] Univ Pau & Pays Adour – IPRA, FR2952, 64000, Pau, France

ARTICLE INFO

Article history:
Received 29 March 2016
Received in revised form
31 March 2017
Accepted 7 April 2017

Keywords:
Air jet impingement
Laminar
Compressible-incompressible modellings
Cylinder
Radial cooling system
Low Mach number
Green aircraft

ABSTRACT

Use of onboard electrical systems is vastly expanding in all aeronautical programs. This means that new cooling systems must increasingly be researched. More specifically, this paper focuses on the cooling of a small turbo engine, represented experimentally as a heating cylinder. This new cooling process designed at Pau University, France, provides quite an original experimental set-up. Given the confined nature of the motion fluid, only experimental measurements of medium physical values such as temperature and velocity can be made. In order to better understand and master such a cooling system, which seemingly could have many industrial applications, two different numerical simulations have been investigated. Because of the confined nature of the motion fluid (V = 6.13 m/s, Mach = 0.018) and the highly complex geometrics, no conclusion can be drawn about the fluid compressibility assumption. Therefore, the incompressible model study is carried out using ANSYS-FLUENT software whereas the compressible model study is conducted with ***elsA*** software. These two computer programs were chosen because of their capacity for double modelling in the hope that some of the major challenges facing the vast research community in the field of low Mach may be addressed. This study is carried out on two different scales. On the coarse scale there are significant similarities between the experimental results and some 2D simulations on the global convective wall heat flux (better than a 10% relative gap). On the smaller scale, code-to-code comparisons of both assumptions (via both codes) show different vortex structures inside the computational domain. Those vortices have a remarkable influence on the inner-wall heat flux's behaviour but their effect is moderate since their average values do not vary much. Though the velocity is relatively low inside the cavity, we will show that the fluid compressibility is of importance in such computations.

1. Introduction - context

There are many reasons to encourage greater use of electrical power: pollution, fuel savings, environmental issues, etc. Increased electrical use appears to be unavoidable. Aerospace industries are not an exception to this rule. The aviation industry is committed to a veritable revolution in the field of energy systems onboard aircraft. Avionics is certain to witness the gradual replacement of hydraulic and pneumatic energy by electricity. The aeronautics sector has long favoured hydraulic energy, which is undoubtedly an efficient solution, but one that presents some major disadvantages. The aircraft's normal movements may cause leaks at components' connections. Moreover, circuits are interdependent and if one circuit fails, the other circuits may not be able to take over. Also, hydraulic fluid is corrosive and flammable. To overcome these problems, a new source of energy must be introduced. Already present in many low-power devices, electricity is currently being studied for devices requiring more power. Recent research work such as GREENAIR (FP7) and the brand-new technological advances achieved on the A380 (Airbus 380) and the Boeing 787 clearly prove that the use of electricity is currently increasing.

As the use of electric-electronic devices increases, more and more efficient cooling systems are required. This study is part of a performance evaluation study of cooling systems for a turbo engine housing, represented experimentally by a small heating cylinder. In 2010 this research project was awarded the Aeronautics Industry Quality Certification label (France) by the three national aeronautical clusters. Funding has continued through 4 consecutive DGCIS-Research Fund grants (France). The original device to be studied can represent any small-sized engine and its potential

* Corresponding author.
E-mail addresses: eric.schall@univ-pau.fr (E. Schall), nicolas.chauchat@univ-pau.fr (N. Chauchat), mathieu.mory@univ-pau.fr (M. Mory).

http://dx.doi.org/10.1016/j.ijthermalsci.2017.04.010
1290-0729/

could be immense in terms of industrial applications. Moreover, it is not restricted to the aeronautics sector. Indeed air jet impingement used for heat treatment is relevant to several engineering applications such as tempering of glass, cooling of turbine blades, drying of some industrial goods and cooling of electronic chips because of the high heat transfer coefficients which occur in the impingement region. Based on the pioneering work by Gardon and Akfirat [9] (along with others) the physics of jet impingement flows has been described in detail. A circular or slot impinging jet on a flat plate serves as a fundamental configuration for many others. Consequently, a multitude of studies have been devoted to these configurations [9–13]. Next, various shapes and nozzles used to generate jets impingement processing systems have been studied and reviewed by many authors. Similarly, applications of the impingement process using various target shapes are gaining popularity in industry. To perform studies of slot jets impinging on complex target shapes, authors often refer to available numerical and experimental findings on flat planes. In Ref. [14], for modelling selection and validation, results from various models are compared against test data for flat-plane jet impingement heat transfer, and a numerical turbulent model based on accuracy is selected to perform jet impingement flows over a cylindrical target surface. Though most reviews focus on the jet impingement cooling of flat surfaces, research into jet impingement on cylinders is expanding [15–20]. Impinging jets on cylinders can be tackled in two different ways: there are authors who consider slot jets to generate jet impingement [19,20] and those who consider circular jets [15–18]. Recently, in Ref. [17], the effect of nozzle shape on jet impingement heat transfer from a circular cylinder has been studied. Experimental and numerical investigations have been carried out to study the effect of different nozzle shapes - circular, square and rectangular. The air jet emerging from the nozzle and impinging on the circular cylinder is unconfined.

In the present study, three main points deserve to be mentioned in comparison to the existing works in the literature dealing with impingement jets. The first one is that the relative low flow magnitude allows us to consider the flow as laminar. Laminar impinging slot jets operating at low Reynolds numbers are often used for cooling of electronics components, especially when these involve either micro-scale fluid phenomena or micro-scale circuit components. The literature on jet impingement under the laminar assumption is very limited. In Ref. [21] different modes of unsteadiness which develop within confined, laminar impinging slot jets of millimetre-scale are considered. Experimental measurements and numerical predictions of different flow characteristics are investigated on a test surface with a constant surface heat flux boundary condition. H. Shariatmadar et al. [22] present a study of heat transfer from an isothermal target surface, which is impinged by an array of air-slot jets. This experimental and numerical study is focused on low Reynolds number ranging from 234 to 470. The variation in heat transfer in laminar wall jets with different initial velocity profiles is studied by Korovkin and Sokovishin [23]. They investigate the effect of the shape of the initial velocity profile on the development of the principle hydrodynamic and thermal parameters. The Reynolds number in their study is about 1400. According to the work by R. Gardon and J.C. Akfirat [24], in regimes of Reynolds number above 3000, impingement flows are essentially turbulent. As the Reynolds number is lower than 3000, the assumption of laminar flow is preferred to turbulent. Moreover, under the laminar assumption the wall heat flux can be easily calculated as follows: $\overrightarrow{q} = -\lambda \overrightarrow{\nabla} T$ where lambda is the thermal conductivity of the fluid and T its temperature. The second point is that we consider the compressibility effects in the numerical part of this work. To the best of our knowledge most numerical studies of jet impingement flows on cylinders are devoid of any compressible effect [9–24]. Though the Reynolds number is relatively high and the flow is air, the assumption of incompressible flow is always made. However, the numerical study is performed for both compressible and incompressible fluid assumptions and the results are then compared. In the present work, the laminar flow field and heat transfer characteristics of radial slot jets impinging onto a heated cylindrical target surface are investigated numerically and experimentally. Air is used as the working fluid with constant properties taking into consideration uniform jet velocity exit impinging on a cylinder target surface of uniform temperature. Slot jet impingement has been studied frequently but relatively little has been published on cases where the cylinder is cooled circumferentially. Only N. Zuckerman et al. [13,14] carried out a numerical investigation on multiple slot jet impingement cooling of circular cylinders, with the nozzles placed circumferentially. They also investigate the effect of conduction in a finite thick cylinder with four impinging radial slot jets. In the present paper, compared to the work of N. Zuckerman et al., the geometry-size and the wall boundaries are different, the jet Reynolds number is one order of magnitude lower and the imposed wall heat flux is approximately seven times lower. An original experimental set-up of such a cooling system device was performed and numerical results compared to the experimental measurement can be considered as the third highlight of this paper.

This paper's main objective is to investigate this type of cooling process. To do this the authors used experience-numerical and numerical-numerical comparisons in two main directions: the first (experiment-numerical) deals with the convective heat flux transfer between the entrance and exit fluids. It should be noted that this comparison is global, i.e. on a large scale. Secondly, a more minute study was conducted through numerical-numerical comparisons to evaluate the effect of compressibility. This step is absolutely necessary since nothing can be deduced from the initial conditions (V = 6.13 m/s, Mach = 0.018). Logically, in the next step, the compressible and incompressible assumptions are tested and compared to each other. Because numerical simulations provide physical solutions at the scaled-particle level, finer comparisons were carried out, particularly on the wall heat flux's behaviour. Comparing experimental and numerical solutions is important, though objective and consistent analysis of results appears to be difficult to carry out even if the comparison is done properly. Indeed, in view of the significant number of parameters that have to be controlled for, there are many possible sources of errors and they are not easy to locate. In order to adjust the discussion frame (section 4), in the following two paragraphs we will attempt to globally identify errors in the experiment and the numerical simulation.

1.1. *From the experimental point of view*

It is well worth pointing out that the confined aspect of the set-up reduces experimental measurement to intrusive measures (cf section 2). Though errors remain possible, no specific study will be conducted on this matter. In this study, measurements of temperatures and flow rate velocity, using respectively thermocouples and flow meters, are supposed to fix the boundary and initial conditions required for the numerical simulation. Finally, it should be noted that the temperature measurement given by thermocouples is always considered as “global” or “average”.

1.2. *From the numerical point of view*

When compared to an experiment, numerical difficulties have to be taken into account differently. Assumptions must be made before computations, for at least one reason: no modelling