Computers & Fluids 101 (2014) 136–154

Contents lists available at ScienceDirect

Computers & Fluids

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

Wall-Modelled Large-Eddy Simulation of a hot Jet-In-Cross-Flow with turbulent inflow generation

S. Bocquet [a,*], J.-C. Jouhaud [a], H. Deniau [b], J.-F. Boussuge [a], M.J. Estève [c]

[a] Centre Européen de Recherche et de Formation Avancée en Calcul Scientifique, 42, Avenue Gaspard Coriolis, 31057 Toulouse Cedex, France
[b] ONERA (French Aerospace Lab), Toulouse, France
[c] Airbus Operations SAS, Rte de Narbonne, Toulouse, France

ARTICLE INFO

Article history:
Received 20 June 2013
Received in revised form 24 April 2014
Accepted 4 June 2014
Available online 19 June 2014

Keywords:
Jet-In-Cross-Flow
Large-Eddy Simulation
Wall modelling
Turbulent inflow generation

ABSTRACT

Hot jets-in-cross-flow are frequently encountered in aeronautics and the accurate estimation of the wall temperature in the jet wake is crucial during the early design of a new aircraft. However, common two-equation RANS models fail at estimating the wall temperature in the jet wake. The use of Large-Eddy Simulation, which seems to be a promising solution at first sight, is not applicable due to its prohibitive computational cost on such large Reynolds number wall-bounded flows. For an affordable cost, we propose a strategy which consists in: reducing the computational domain to a small region around the phenomenon of interest (RANS-LES embedded approach), perform a Wall-Modelled Large-Eddy Simulation (WMLES) in the reduced domain and generate a turbulent inflow at the reduced domain inlet. The test case selected is a hot Jet-In-Cross-Flow experimentally studied by Albugues (2005) [1]. We simulate the real geometry of the wind-tunnel model, which imposes strong constraints on the inflow generation and numerical method. It is shown that an advanced inflow generation, combining a stochastic velocity fluctuation injection and a dynamic forcing term (Laraufie et al., 2011) [17], is mandatory to obtain a realistic turbulent flow upstream of the jet. In the jet wake, the wall temperature estimated by the WMLES agrees well with the experimental measurements.

1. Introduction

Jet-In-Cross-Flow (JICF) are commonly encountered in aeronautics and their application range from turbine cooling in jet engines to flow control and hot air exhaust in external aerodynamics, to state a few of them. JICF for hot air exhaust is the subject of this study and the application targeted here is the anti-icing system of aircraft engine nacelles (see Fig. 1(a)). As shown in Fig. 1(b), the anti-icing system consists in a circulation of hot air in the nacelle leading edge, which heats the leading edge and prevents ice accumulation. The hot air then enters in a plenum before exiting and mixing with the main flow surrounding the aircraft. It appears that downstream of the jet, the hot air impacts the composite materials forming the engine nacelle. This composite material is thus submitted to repeated thermal stresses which can lead to abnormal fatigue and finally to structural damages. To prevent these damages, the composite materials are protected by thermal shields whose size should be minimized to avoid useless weight. As a result, an accurate description of the wall temperature field in the jet wake is of special concern from the industrial point of view.

During aicraft design, the Reynolds-Averaged Navier Stokes (RANS) approach is suitable for simulating most applications, with the advantage of a moderate computational cost. However, common one or two-equation RANS models have been shown to fail at predicting the wall temperature field behind a hot JICF, as shown by Albugues [1], Jouhaud et al. [2] and Duda [3]. Duda also evaluated the suitability of Unsteady RANS (URANS) to simulate such JICF, without clear improvements of the results. The reason identified for the failure of (U) RANS is the presence of several large scale coherent motions with broad spectral content, which determine the development of the jet wake and the wall temperature distribution downstream of the JICF.

There is general agreement that LES is well suited for the simulation of JICF [2,4]. However it is known that LES involves a prohibitive computational cost as soon as large Reynolds number wall-bounded flows are concerned, such as the JICF described above. This computational cost is due to the presence of very small streaky turbulent structures in the inner layer of the boundary layer, which require a very fine mesh to be properly captured. According to the estimates of Choi and Moin [5], the LES of a

* Corresponding author. Tel.: +33 6 72 68 60 69.
E-mail address: sbocquet@gmail.com (S. Bocquet).

http://dx.doi.org/10.1016/j.compfluid.2014.06.015
0045-7930/

(a) (b)

Fig. 1. (a) Hot air exhaust from the anti-icing system on a A380 engine. (b) The different components of the anti-icing system.

turbulent wall-bounded flow involves a number of computational cells proportional to $Re_x^{1.9}$ where $Re_x = \rho_\infty U_\infty x/\mu_\infty$ is the Reynolds number based on the distance x from the leading edge.

To alleviate this expensive computational cost, hybrid RANS-LES methods [6] attempt to use RANS in the boundary layer and LES elsewhere. Among the hybrid RANS-LES methods, Wall-Modelled LES (WMLES) [7,8] consists in resolving the turbulent structures in the outer layer of the boundary layer, while modelling the effect of the smallest structures underneath. By modelling these very small structures, the number of cells needed becomes proportional to $Re_x^{0.4}$ according to Chapman [9],[1] allowing a much smaller number of cells than the one required by a wall-resolved LES.

WMLES has been successfully applied to JICF by Jouhaud et al. [2] and Hallez et al. [4]. In addition to the WMLES approach, we will focus on two specific points: (1) further reducing the computational cost by limiting the computational domain to a small region around the JICF, which is called the 'RANS-LES embedded approach' and (2) generating appropriate turbulent inflow at the inlet of the WMLES domain. Both the WMLES approach and the inflow generation are implemented in the elsA software [10], which is a multi-block structured compressible flow solver, capable of massively parallel simulations and used by EADS and SAFRAN in their design process.

The problem of inflow generation for LES has been the subject of several studies. However, inflow generation for WMLES has focused very minor attention and very few work exists on this subject. Thus, the investigation of inflow generation for WMLES constitutes the original part of this work.

This study is structured as follows: (1) the JICF configuration is described; (2) The strategy chosen to tackle the LES of large Reynolds number wall-bounded flows, based on WMLES, the RANS-LES embedded approach and turbulent inflow generation, is presented; (3) The RANS modelling; (4) The LES modelling are then described; and (5) Results are discussed, starting with the effect of inflow generation upstream of the JICF. Then, WMLES results in the jet wake are compared to experimental results. Finally velocity spectra are analyzed and the numerical method limitations are discussed.

2. The Jet-In-Cross Flow configuration

The configuration studied is not the real anti-icing system but the representative wind tunnel model investigated experimentally by Albugues [1]. Fig. 2(a) shows the wind tunnel test section, of dimension $5 \times 1.4 \times 1.8$ m respectively in the streamwise, spanwise and vertical directions. An airfoil of $C = 0.7$ m chord is fixed between the two lateral walls and contains internal equipments able to generate the JICF (see Fig. 2(c)). The use of an airfoil is motivated by the objective of reproducing the wall pressure distribution that takes place on a real aircraft engine nacelle. Hot air at a total temperature of 363 K is supplied inside the airfoil through two symmetrical pipes at a given mass flow rate q_j. The hot air then mixes in a plenum located right below the ejector grid of thickness $d = 2$ mm. The hot air finally exits through a square exhaust hole made in the ejector grid and interacts with the wind tunnel main flow at ambient temperature, forming the JICF. It should be noted that the plenum walls are cooled by circulation of cold water in small pipes surrounding the plenum. Thus the total temperature of the hot air right below the exhaust hole has decreased to a value of about 353 K [1]. The main flow velocity is $U_\infty = 47.25$ m s^{-1} at a static temperature $T_\infty = 295$ K, leading to a Mach number $M_\infty = 0.14$ and a Reynolds number $Re_D = \frac{\rho_\infty U_\infty D}{\mu_\infty} = 93,000$, with $D = 30$ mm is the exhaust hole dimension and ρ_∞ and μ_∞ are respectively the main flow density and molecular viscosity. Expressed using the exhaust hole dimension, the domain measures $[-83D, 83D] \times [-23D, 23D] \times [-30D, 30D]$ respectively in the streamwise, spanwise and vertical directions. Transition is triggered upstream of the ejector grid so that the boundary layer is fully turbulent when it reaches the grid.

The Reynolds number, Mach number, wall pressure distribution and the temperature difference between the hot and cold flows, here equal to $\Delta_T = T_j - T_\infty = 57$ K, constitute a set of similarity parameters that characterize the JICF dynamics. In addition to these, two important similarity parameters can be further identified: the ratio of momentum between the hot and cold flow $C_R = \frac{\rho_j U_j}{\rho_\infty U_\infty}$ and the ratio of main flow displacement thickness by exhaust hole dimension $\frac{\delta_1}{D}$. Here $C_R = 0.69$ and $\frac{\delta_1}{D} \approx 17 \times 10^{-3}$, the latter being measured just upstream of the exhaust hole from the RANS computation described later.

The X, Y, Z coordinates and U, V, W velocity components respectively stand for the streamwise, spanwise and vertical directions in the global frame of reference. The origin of this frame of reference is located at the middle of the exhaust hole downstream edge. x, y, z coordinates and u, v, w respectively denote streamwise, wall-normal and spanwise directions in the local boundary layer frame of reference.

3. A strategy for the Large-Eddy Simulation of large Reynolds number wall-bounded flows

3.1. The Wall-Modelled Large-Eddy Simulation

As seen in the introduction, the computational cost of a wall-resolved LES would be very expensive on this JICF. Indeed the Reynolds number based on the airfoil chord is $Re_c = 2.2 \times 10^6$. To fulfill the criterion $\delta_x^+ = 50, \delta_y^+ = 1$ and $\delta_z^+ = 15$, we estimate that at least 300×10^6 cells would be necessary to discretize the small embedded domain shown in Fig. 3 with a conventional full-matching structured mesh. Here the superscript $(.^+)$ designates dimensions in wall units, that is:

[1] Choi et al. estimate the number of cells proportionnal to Re_x for $Re_x > 10^6$.