Computers and Fluids 152 (2017) 134–156

Contents lists available at ScienceDirect

Computers and Fluids

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

Large-eddy simulation of an air curtain confining a cavity and subjected to an external lateral flow

J. Moureh*, M. Yataghene

Refrigerating Process Engineering Unit, Irstea Antony, 1 rue Pierre-Gilles de Gennes – CS10030, 92761 Antony Cedex, France

ARTICLE INFO

Article history:
Received 3 May 2016
Revised 11 April 2017
Accepted 17 April 2017
Available online 25 April 2017

Keywords:
LES
Airflow
Air curtain
Perturbed jet
Turbulence
CFD

ABSTRACT

In this work, large eddy simulations of a plane jet confining a cavity and subjected to an external lateral flow (ELF) were performed. Alongside and in a complementary manner an experimental study was carried out on a scale down model (1:5) representing a generic configuration of a display case, using LDV and PIV techniques to investigate the air flow characteristics. The jet behavior is examined using the mean velocity field, the turbulence characteristics, coherent structures by using the Q-criterion, and Strouhal number. The transport of a passive scalar was also considered to illustrate the dynamic interactions between the jet and its surroundings and thus, to better investigate the effect of the ELF on the transient jet behaviour. The statistical LES results will be analyzed and validated with the experimental results and a good agreement was achieved. The results are also compared with the standard k-ε model in order to achieve a critical evaluation of this model widely used in the studied configuration. Between the two numerical methods employed, LES performs better than RANS in predicting the jet behavior and jet characteristics. In addition, LES proves its ability to predict not only self-similarity properties, but also provide the pattern of large-vortex structures and their temporal evolution, and hence, successfully resolves the transient mixing process between the jet and its surroundings with and without external perturbation.

1. Introduction

In industrial and commercial fields, air curtain devices formed by a plane jet or a combination of jets, are widely used as dynamic barriers instead of physical barriers to create opened and conditioned spaces with a control of ambient parameters such as temperature, humidity and contaminants. The aim of air curtain is to reduce and to control heat and mass transfer between the protected area and the outside through the air jet, while to provide an easy access for people and vehicles. One of the relevant applications is the vertical open Refrigerated Display Cabinet (RDC) widely used in supermarkets. In a typical RDC, the air jet flows from nozzle inlet located at the top front to a nozzle exit located at the bottom front of the case, acts as a thermal barrier between the warm ambient air and the chilled compartment. Due to their design, RDC are very sensitive to ambient conditions and they are considered as the weakest link of the cold chain [1]. This could be explained by the fact air curtains are easily disturbed by the outside ambient air, which in addition results in higher temperature rise and more power consumption.

Many authors [2–9] have shown computational fluid dynamics (CFD) to be a valuable tool to rapidly provide design options to improve airflow within display cabinets. In their studies, the authors use the CFD models to optimize RDC design or to minimise energy losses through air curtain by testing the influence of the main factors which include the dimensions of the nozzle inlet, the nozzle exit, the length of air curtains, the initial velocity, turbulent intensity and temperature of air curtains and the temperature and velocity of the ambient environment. One can mention the simplified numerical approach conducted by Cortella et al. [3] who used large eddy simulation (LES) based on streamfunction-vorticity to investigate temperature and velocity distributions in RDC. However, in this modelling approach, the streamfunction patterns can only give a qualitative idea on the flow field and no details concerning turbulence characteristics can be obtained.

However, it's worth to notice that the validation of the majority of CFD models reported in the literature [2,3,7,8] are obtained by comparisons with test results limited to temperature and humidity (no velocity measurements were performed). Even when velocity measurements have been performed, they are limited to some velocity profiles [9] which is not enough to validate complex 3D airflow patterns [10]. Thus, there is a need to use of more advanced non-intrusive techniques like PIV and LDV in order to

* Corresponding author.
E-mail address: jean.moureh@irstea.fr (J. Moureh).

http://dx.doi.org/10.1016/j.compfluid.2017.04.013

Nomenclature

C	Dimensionless mass fraction [-]
C_p	Specific heat of air [J kg^{-1} K^{-1}]
D_h	Hydraulic diameter of inlet [m]
e	Nozzle inlet width [m]
H	Height of the cavity [m]
I	Turbulence intensity ($I_i = \frac{\sqrt{\overline{u_i^2}}}{U_t}$) [%]
k	Kinetic energy of turbulence [m^2.s^{-2}]
Ky	Jet-spreading rate [-]
Ku	Velocity decay rate [-]
L	Nozzle inlet length [m] [m]
p	Static pressure [Pa]
Re	Reynolds number, $Re = \rho U\, e/\mu$ [-]
T	Temperature [K]
$\overline{u_i' u_j'}$	Reynolds stresses component [m^2.s^{-2}]
U_i, u'_i	Mean and fluctuating velocity component in x_i direction [m.s^{-1}]
U, V W	Mean velocity components in x, y and z direction
u_i	Instantaneous velocity component [m.s^{-1}]
U_{rms}	Streamwise RMS velocity $= \sqrt{\overline{u'^2}}$ [m.s^{-1}]
W_{rms}	Lateral RMS velocity $= \sqrt{\overline{u_z'^2}}$ [m.s^{-1}]
$y_{1/2}$	Jet half-width, calculated at the y-location at which $U(x) = ½\, U_m\, (x)$
x,y,z	Streamwise (x), spanwise or transverse (y) and lateral (z) directions in Cartesian Coordinates [m]

Greek symbols

ε	Turbulent dissipation rate. [m^2.s^{-3}]
δ_{ij}	Kronecker symbol
μ	Laminar dynamic viscosity [Pa.s]
λ	Thermal conductivity [W.m^{-1}.K^{-1}]
μ_t	Turbulent viscosity [Pa.s]
ν	Kinematic viscosity [m^2.s^{-1}]
ρ	Density [kg.m^{-3}]

Subscript

0	relative to inlet boundary condition
a	ambient
lf	lateral flow
m	maximum
min	*minimum*
sgs	sub-grid scale
–	time-averaged, filtred field
⊥	normal
i, j, k	relative to coordinate system
x, y, z	relative to coordinate system

Acronyms

ELF	**E**xternal **L**ateral **Flow**
K–H	Kelvin–Helmholtz
LDV	**L**aser **D**oppler **V**elocimetry
PIV	**P**article **I**mage **V**elocimetry
RDC	**R**efrigerated **D**isplay **C**abinet
RMS	**R**oot **M**ean **S**quare

obtain high enough resolution to characterise air flow patterns and velocity profiles and turbulence levels. The obtaining of such results allows a better understanding of airflow characteristics and also improves the quality of CFD validation.

According to typical RDC dimensions and the corresponding jet flow rates, the Reynolds number defined as $Re = \rho U\, e/\mu$ is about 5000 and the typical air curtain lengths are about ten jet widths [11]. Therefore, the air curtain flows reside in the transitional flow region, which is not well understood and little information was available concerning the velocity profiles. In addition, the transitional flow is more complex to predict since classical turbulence model RANS are more adapted to fully developed turbulence. Experimental studies [12–15] demonstrate that all turbulent free jet flows have similar structures i.e. the presence of vortex rollup or counter-rotating vortices due to the Kelvin–Helmholtz (K-H) instabilities in two shear layers which are separated by a potential core region *beyond* which the flow eventually reaches a fully developed self-similar state.

Based on the literature, it can be seen that the majority of these works listed above takes in consideration only an idealized scenario in which no perturbation affects the dynamic behavior of the air curtains. However, these facilities generally used in open spaces like supermarkets, are very sensitive to external perturbations generated by human activities such as pressure difference due to door opening, parasitic air draughts, action of air conditioning system, etc. Such perturbations may strongly affect the stability, the transfer mechanisms and therefore the efficiency of air curtain due to a lack of confinement and an increase of energy consumption [16]. D'Agaro et al. [7] indicated that the incidence of external air currents on air entrainment needs further investigations. Therefore, the flow characteristics, efficiency and performance are yet to be understood, especially with a consideration on external perturbations, which is more close to practical industrial applications

Some fundamental studies have also been extended by number of researchers to jets discharged in no-quiescent surroundings with "co-flow" [17,18] and "cross-flow" [19,20]. However, excepting our earlier studies made recently [21,22], and to the best of our knowledge, the flow configuration of a perturbed air curtain resulting from an aerodynamic interaction with an external "lateral flow", has not been yet studied in detail either experimentally or numerically. In [21] a global numerical model based on the RANS approach build with RSM was performed in order to predict air fluxes exchanged between the air curtain, the ambient and the cavity, while in [22], a comparative study was performed between two RANS models, the RSM and the k-ε model.

The aim of this research work is to experimentally and numerically characterize the dynamic behavior of an air curtain used to confine a rectangular cavity and subjected to external perturbation. This study is performed on a reduced-scale model representing an idealized configuration of a RDC, which is formed by an isothermal simple jet plane, and an investigation of the influence of the main parameters related to the jet, and the lateral perturbation. This allows performing more fundamental study on the physics of an air curtain, while avoiding more case-specific geometries used by many authors [2,4,7,23–26]. Velocity measurements are performed by means of LDV and PIV techniques. The main purpose is to acquire reliable data of the physics of the air curtain development and transfer mechanisms with and without external perturbation.

The study focuses the near field region ($x/e < 10$) due to its strong relevance to RDC which includes the transition zone where strong interactions are expected between the jet core, cavity and external lateral flow. The resulting jet flow topology is complicated since it is often the combination of free turbulent shear dominating-flow, near-wall effect, secondary-recirculating flow including curvature effects within cavity and anisotropic vortical structures. An undeniable difficulty arises through dynamic lateral interactions with an external flow implying an additional shearing effect between the jet and its surroundings in the lateral direction, which can strongly affect the jet development and the inherent K-H instabilities.

Concerning the simulation of turbulent flows, different approaches can be attempted. The direct numerical simulation (DNS) is the most accurate since all the scales of motion are resolved