Available online at www.sciencedirect.com

ScienceDirect

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

Numerical and experimental analysis of the air flow distribution in the cooling duct of a display cabinet

Sergio Marinetti [a,*], *Giovanna Cavazzini* [b], *Igor Lauri* [c], *Samuele Testa* [a], *Silvia Minetto* [a]

[a] *National Research Council, Construction Technologies Institute, Corso Stati Uniti, 4, 35127 Padova, Italy*
[b] *University of Padova, Department of Industrial Engineering, via Venezia 1, 35131 Padova, Italy*
[c] *Arneg S.p.A., via Venezia 58, 35010 Campo San Martino, PD, Italy*

ARTICLE INFO

Article history:
Received 11 November 2013
Received in revised form
16 February 2014
Accepted 18 February 2014
Available online 27 February 2014

Keywords:
Particle Image Velocimetry (PIV)
Display cabinet
Computational fluid dynamics
Propeller fan
Heat exchangers
Air flow pattern

ABSTRACT

This work presents an experimental and numerical analysis of the flow distribution in the cooling duct of a commercial refrigerated display cabinet. The analysis is carried out on a channel mock-up, in which the air is forced through the evaporator by two fans. A steady-state isothermal numerical analysis of the flow distribution is performed with the commercial code ANSYS-CFX and the k-epsilon method is used to model the turbulence. The numerical results are compared with the 3D air velocity fields, taken on three planes placed at different heights inside the duct, by the stereoscopic Particle Image Velocimetry (PIV) technique. Once validated, the model allows a better comprehension of the flow maldistribution source and its effect on the velocity field at evaporator exit.

Analyse numérique et expérimentale de la distribution de l'écoulement d'air dans la conduite de refroidissement d'un meuble frigorifique de vente

Mots clés : Vélocimétrie par images de particules (PIV) ; Meuble frigorifique de vente ; Dynamique numérique des fluides ; Hélices de ventilateurs ; Echangeurs de chaleur ; Schéma d'écoulement d'air

---

[*] *Corresponding author*. Tel.: +39 (0)49 8295746; fax: +39 (0)49 8295728.
E-mail address: sergio.marinetti@itc.cnr.it (S. Marinetti).

http://dx.doi.org/10.1016/j.ijrefrig.2014.02.009

1. Introduction

In the last fifteen years Computational Fluid Dynamic (CFD) has been widely used to simulate open refrigerated display cabinets (ORDC). Modelling has been mainly intended to describe heat and mass transfer through air curtain, trying to evaluate interaction between the display cabinet and the surroundings (Navaz et al., 2002). Early in 1997 Stribling et al. (1997) were mostly interested in minimising energy consumption and improving cold store comfort by analysing the overall heat loss from the case. The same general purpose is still maintained in more recent papers, with special emphasis on keeping product temperature below a critical value consequently preserving its quality (Foster et al., 2005). 2D simulations, considering the middle section as representative of the air curtain behaviour, are often met in the previous literature due to their simplicity and low CPU times. It is worth mentioning Cortella et al. (2001), who performed a CFD simulation using a Large Eddy Simulation (LES) model for the air flow in a double-curtain display case, Ge et al. (2010), who integrated the CFD model with a cooling coil sub-models and Yu et al. (2009), who developed the correlation model of the Thermal Entrainment Factor (TEF) for typical single and double air curtain in a vertical ORDC based on CFD simulation. 3D models were introduced to study the effects of a uniform flow of ambient air across the front of the case with a velocity of 0.2 m s^{-1}, as prescribed by the UNI EN ISO 23953 Standard (Stribling, 1997). D'Agaro et al. (2006) carried out 3D CFD parametric studies to evaluate the influence of longitudinal ambient air movements and extremity effects due to the display cabinet length on the ORDC low temperature performance. Gaspar et al. (2010) presented a 3D modelling of the physical-mathematical phenomena occurring in an open refrigerated display cabinet, under different magnitude values of the ambient air velocity. Besides these external factors, there could be an internal source contributing to the uneven curtain performance. The velocity profile at the Discharge Air Grille (DAG), which is normally imposed as a boundary condition in numerical models, can be affected by the combined effect of the fans. Marinetti et al. (2012) experimentally studied the flow field inside a cooling duct by means of 3D PIV. They highlighted the development of a 3D secondary flow in the region between the fans discharge and the evaporator entrance. This secondary flow determined a mal-distribution of the mass flow rate in the channel, with two main flows localized near the side walls, and a flow blockage in the central part of the duct. In the present work, a 3D CFD model of the air channel studied in Marinetti et al. (2012) is developed in order to simulate and validate the experimental evidence attained with the PIV technique.

2. The case study and experimental results

The analysed system reproduces the cooling duct of a horizontal open type cabinet, with the evaporator located in the bottom of the chest. The duct, made of plexiglass to allow the optical access for laser and cameras, is represented in Fig. 1a. Two "almost-horizontally" mounted axial fans force the air through a finned-tube evaporator. More details about the experimental setup can be found in Marinetti et al. (2012). The air flow distribution inside the channel was mapped by the PIV technique. The measurements were performed on three different planes, as indicated in Fig. 1b, placed at $1.9 \cdot 10^{-2}$ m, $3.8 \cdot 10^{-2}$ m and $5.7 \cdot 10^{-2}$ m from the bottom of the duct. The experimental results were presented as intensity images and arrow field plots. For instance, Fig. 2a shows the velocity component along the mainstream direction (x axis in Fig. 1a, where a positive velocity is toward the fan) on Plane 2, while Fig. 2b represents the arrow field plot of the velocity projections on the same plane. In the region depicted by the white band, no measurements are available due to the presence of the evaporator, which is not transparent thus preventing the PIV analysis.

3. Three-dimensional simulations

3.1. *The numerical model*

The entire cooling duct is modelled with an unstructured grid of more than 3,100,000 nodes and numerically analysed by means of the commercial software package ANSYS CFX 14.0. The turbulence is modelled by a k-epsilon model and the computation is carried out adopting a high resolution advection scheme.

As regards the evaporator, the detailed description of the evaporator micro-channels in the discretized domain would be extremely time-consuming due to the enormous request of the mesh nodes as a consequence of the existence of two different geometric scales (a millimetric scale for the evaporator and a metric scale for the cooling duct). To avoid time-consuming analyses, the evaporator fins are not physically

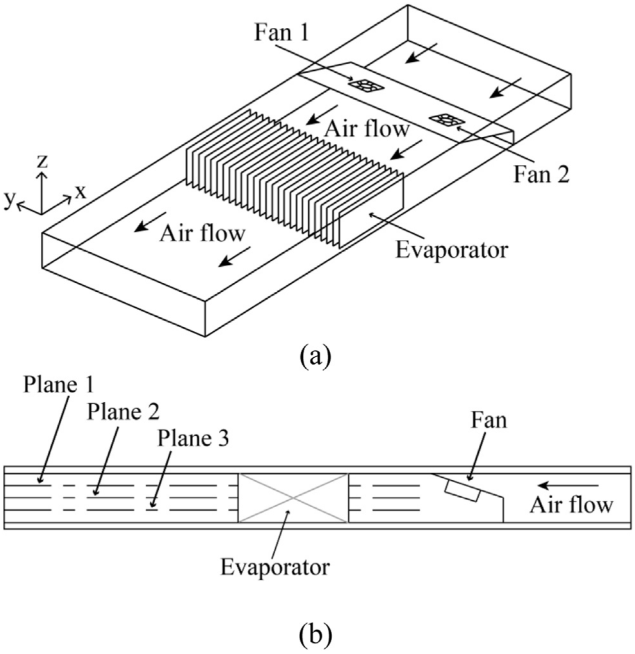

Fig. 1 – The cooling duct (a); the laser measurement planes (b).