Applied Thermal Engineering 87 (2015) 24–33

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research paper

Numerical modelling and validation of the air flow maldistribution in the cooling duct of a horizontal display cabinet

Antonio Rossetti*, Silvia Minetto, Sergio Marinetti

Construction Technologies Institute, National Research Council, Corso Stati Uniti 4, 35127 Padova, Italy

HIGHLIGHTS

- A CFD model of a cooling duct affected by mal-dist. is presented and validated.
- Air-side maldistribution is a challenging phenomenon for CFD.
- The sensitivity to boundary conditions and turbulence models is discussed.
- Validation of the velocity field is carried out using PIV measurements.

ARTICLE INFO

Article history:
Received 19 September 2014
Accepted 8 April 2015
Available online 24 April 2015

Keywords:
Refrigeration
Display cabinet
Computational fluid dynamics
Particle image velocimetry

ABSTRACT

Air maldistribution strongly affects the performance of Open Refrigerated Display Cabinets (ORDC). Computational Fluid Dynamics (CFD) can be a useful approach to model both causes and effects of air-side maldistributions. One of the main drawbacks is the complexity of the resulting model and the high computational costs. In this paper, a numerical model of the air channel of an ORDC is presented and validated. The numerical set up was oriented to reduce the computational load, while maintaining a good agreement with experimental PIV data. Results highlighted the great importance of the correct set up of the inlet boundary and of the evaporator volume description. In addition, the modelling of the flow transition from turbulent to laminar inside the evaporator volume was proven to be critical to obtain a correct wake description downstream the evaporator.

1. Introduction

Under the pressure of stricter environmental constraints imposed by legislation and market pressure, commercial refrigeration is undergoing a tough process aiming at improving performances and energy efficiency of subsystems, such as display cabinets, refrigeration units, etc, as well of the entire supermarket system.

Open refrigerated Display Cabinets (ORDC) have been the focus of a large research activity to improve the insulation of the goods from the surrounding ambient and reduce the energy consumption. In order to obtain such results, Computational Fluid Dynamic (CFD) has been extensively used to model both the external and internal flow in ORDC.

Many authors adopted 2D simulations to obtain a suitable thermo-hydraulic model of the ORDC. Ge [1] and Gaspar [2], for example, studied the ORDC by means of a 2D CFD model coupled with an analytical model of the heat exchanger. On the other side, 3D effects were demonstrated to be important to obtain detailed information in both the external [3] and internal flow [4]. Indeed, air mal-distribution can affect the insulation performance of the curtain and the heat removal in the evaporator. Furthermore, the presence of mal-distribution downstream the evaporator can affect the air distribution at the cabinet back panel (BP) in vertical cabinets, compromising the heat removal from the goods.

For these reasons, the development of a validated procedure to model the 3D flow inside the evaporator channel of an ORDC can be a valuable tool to improve actual design. Correlation based models (such as the ε-NTU method), capable of handling to some extent the effects of these mal-distributions, had been proposed to predict the performance of heat exchanger under non uniform inflow conditions. Koern et al. for example, developed in Ref. [5] an analytical model of a fin and tube evaporator in order to discuss the effects of maldistributions. The model idealized both the condenser and the evaporator as composed by two straight tubes, and discussed the effects of uneven distribution between the tubes of: liquid–vapour

* Corresponding author.
E-mail address: antonio.rossetti@itc.cnr.it (A. Rossetti).

http://dx.doi.org/10.1016/j.applthermaleng.2015.04.032
1359-4311/

phase, refrigerant side distributed losses and air velocity. The same authors in Ref. [6] used the model to compensate the maldistributions by controlling the superheating of each individual tube of the evaporator. Mao et al. [7] used a correlation based model to discuss the effect the air side maldistributions on a multi-louvered condenser, using four predefined airflow profiles: uniform, paraboloid, saddle and linear. Despite these approaches can give useful information on the effects of maldistribution their capability to model actual maldistribution is still limited.

On the other side, CFD has proved to be an effective tool for the study of heat exchangers [8]. Many numerical researches focus on a single fin model to optimize the evaporator design, assuming implicitly uniform inlet condition and the absence of both refrigerant and air maildistributions. For example Karmo et al. [9] designed an inclined fin and tube exchanger using CFD simulation of fins passage under symmetry/periodic assumption. Bhuiyan A.A. et al. [10] tested the performance of a fin and tube heat exchanger under transition conditions using similar simplifications, while Bilirgen H. et al. [11] tested the sensitivity of a finned tube element performance to the main geometrical parameters of the fin. While these studies provide useful information during the preliminary design stage, they cannot assess the impact of the actual installation on the evaporator performance. In order to simulate the interaction between the air path and the evaporator the whole system has to be analysed. For example, Moukalled et al. [12] used 3D CFD code to model and optimise the air flow on a rooftop air conditioning unit. The model included all the main causes of air maldistribution such as the casing and the fans, and heat exchange was solved modelling all the fins of both evaporator and condenser units.

One of the main drawbacks of the correlation-based approach is the lack of flexibility in the description of complex 3D fields. Furthermore, the velocity field, which should be a result of the method, had to be provided as an input of the model. On the other side, the full 3D CFD description proposed by Moukalled [12] relies on huge computational efforts, requiring long time and expensive hardware to get results.

The present work focuses on the development of a validated CFD model of the internal channel of an ORDC, capable of good accuracy and reduced computational load. In order to get accurate results, two main points should be addressed: the fans, which are the main cause of the air mal-distribution, and the proper set up of the evaporator numerical model. In this work both these point are discussed.

Inlet boundary was then set immediately downstream the fans outflow. Fans were not included into the model for two main reasons. First, the inclusion of fans drastically increases the computational load of the model. Secondly, fans geometry is usually not available to the cabinet designer, as commercial fan are generally employed. In Ref. [13], Marinetti et al. already discussed the importance of this boundary condition and proposed Particle Image Velocimetry (PIV) measures at the exit of the fan in free air, in order to supply representative boundary for the numerical model.

In this paper, the boundary field was measured recreating a short extent of the channel around the fan, improving significantly the model accuracy.

The evaporator geometry was simplified using the equivalent resistance approach, as in Ref. [13] and [14], to obtain a good compromise between accuracy and model complexity (computational time). Detailed analyses of a single fin passage were used to tune the parameters of the evaporator. The correct modelling of the laminar–turbulent transition are proved to play an important role to obtain good agreement with experimental data when the evaporator is accounted.

Numerical results will be discussed in comparison with PIV experimental data.

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 Plexiglas 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. The channel was tested at room temperature and all the experimental facility, including the evaporator, was in thermal equilibrium with the ambient. The air flow distribution inside the channel was mapped by PIV technique.

The PIV system used in this study is commercially available from LaVision GmbH and permits the fulfilment of 2D and 3D air flow measurements over fields of view (FOVs) up to about 50 cm $\times$ 50 cm. The apparatus mainly consists of a green double-pulsed Nd:YAG laser with 15 Hz repetition rate and 125 mJ energy (NANO-L-125-15 PIV – Litron Lasers), two thermoelectrically cooled cameras with 4 M pixel CCD sensor (Imager ProX – PCO Imaging), and a Programmable Timing Unit (PTU) for the devices synchronization. For seeding the flow of interest, olive oil particles of uniform size were produced by a self-made Laskin generator. Seeding particles diameter was in the order of **0.2** μm. The data processing was carried out with DaVis software package (LaVision GmbH) and a proprietary Matlab code.

The measurements were performed on three different planes, as indicated in Fig. 1b, placed at $1.9\cdot10^{-2}$ m, $3.8\cdot10^{-2}$ m and $5.7\cdot10^{-2}$ m from the bottom of the duct. Each plane was subdivided in 20 FOVs. For each FOV, in order to have a reliable representation of the mean local flow, 35 images were grabbed at a frequency of 2 Hz, and then averaged together.

Measurements uncertainty results from both the calibration and the correlation errors. The PIV system was calibrated using a two-step calibration procedure. Firstly, the calibration was performed on the 3D target and then it was refined through a self-calibration, made on the particles themselves. The RMS of the calibration error was estimated to be about 0.1 pixels.

The uncertainty due to the correlation procedure was estimated by processing synthetic images produced by deforming experimental images of the particle by a predefined displacement. The RMS of the correlation algorithm was estimated to be 0.15 pixel.

Correlation and calibration errors can be combined as the camera reciprocal position is known, obtaining an in-plane (xy plane) uncertainty of 5–3% and an out of plane (z direction) uncertainty of 8–5 %, as the time intervals between the laser shots was tuned to obtain displacements in the range 5–8 pixels. More details about the experimental setup can be found in Marinetti et al. [15].

Results are reported in Fig. 2, representing average x component of velocity and the turbulent kinetic energy in the middle plane (plane 2) without and with the evaporator. The white band in Fig. 2b and d correspond to the evaporator, as it was not transparent, thus preventing PIV analysis. Both the configurations present a significant maldistribution of the air, as results of the fans positioning and rotation. Both the fields (Fig. 2c and d) present also high turbulent kinetic energy in the proximity of the fans discharge, as the almost-horizontal position of the fans let the flow interacts violently with the bottom of the channel causing high instability. The evaporator damp almost the turbulence, while when it is absent turbulence evolve downstream along the jet–wake interface.

3. Numerical domains and grids

The channel with and without evaporator was modelled. Structured grids were used in both cases. The duct without the evaporator consists in one domain, while, when considering the