Applied Energy 89 (2012) 150–155

Contents lists available at ScienceDirect

Applied Energy

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

Computational fluid dynamic investigation of liquid rack cooling in data centres

Ali Almoli [a], Adam Thompson [a], Nikil Kapur [a], Jonathan Summers [a], Harvey Thompson [a,*], George Hannah [b]

[a] Institute of Engineering Thermofluids, Surfaces & Interfaces (iETSI), School of Mechanical Engineering, University of Leeds, Leeds, LS2 9JT, United Kingdom
[b] Airedale International Air Conditioning Ltd, Leeds, United Kingdom

ARTICLE INFO

Article history:
Received 12 July 2010
Received in revised form 21 January 2011
Accepted 3 February 2011
Available online 25 February 2011

Keywords:
Data centres
Liquid cooling
Heat transfer
CFD modelling

ABSTRACT

Relying on thermal air management in a data centre is becoming less effective as heat densities from the Information Technology (IT) equipment continue to rise. Direct liquid cooling is more efficient at transferring the waste heat, but requires liquid loops passing as close as possible to the heat source. A new Computational Fluid Dynamics (CFD) strategy is developed for data centre scenarios where a liquid loop heat exchanger is attached at the rear of server racks (back doors), which can avoid the need to separate the cold and hot air streams in traditional hot/cold aisle arrangements. The effectiveness of additional fans in the back door heat exchangers is investigated using the three-dimensional CFD model of a simplified three-aisle, six-rack data centre configuration.

1. Introduction

The rapidly increasing energy demand of data centres is presenting industry and governments with an energy supply problem. It is clear that the current roadmap of higher density servers with no radical changes in computing technology on the horizon will lead to year-on-year increases in energy requirements since data centre power consumption has doubled in the last 5 years and is likely to double again in the next 5 years to over 100 billion kW h [1]. Within the UK, the issue of thermal management in data centres is becoming critical since power supply is becoming restricted in key data centre locations such as London Docklands, The Thames Valley and Manchester [2].

There are two main inefficiencies leading to such enormous energy requirements: the Information Technology (IT) hardware inefficiencies and the cooling requirements, each accounting for roughly 40% of the total energy usage, with the result that each kW h of energy for data processing requires a further kW h for cooling. The energy-efficient design of data centres is a truly multi-disciplinary problem. IT load inefficiencies can be addressed by improved semiconductor technologies [2] and server virtualisation, while cooling of the electronics in data centres can be achieved in a number of ways, by far the most popular at present being via cold air, but there is a trend to adopt direct liquid cooling [3] of the servers by, for example tube and fin heat exchangers attached to the back of the server rack or, yet to be fully utilised, dielectric liquid immersion cooling and on-chip spray cooling [4].

In this paper we focus on thermal air flow management and cooling in data centres, issues which are particularly challenging since Computer Room Air Conditioning (CRAC) systems have to maintain temperatures and humidity levels in narrow bands in order to avoid catastrophic data losses due to over-heating servers, hygroscopic dust or electric discharge failures. Strategies are usually based on the separation of hot and cold air via a layout of hot and cold aisles, but efficiencies can be gained with either of the two aisles contained. The CRAC units usually supply cold air into data centres through raised floor tiles and the cold air passes through the server racks, cools the electronic equipment and emerges from the back of the servers as a hot air stream. Computational Fluid Dynamics (CFD) is increasingly being used to improve air flow design in such systems [5,6]. The objectives of this paper are to: (i) develop a new CFD modelling strategy to investigate the effectiveness of liquid loop heat exchangers mounted to the rear of the IT server racks, referred to as a back door cooler, and (ii) apply the CFD model to a simplified three-row, six-rack high-density data centre scenario where the back door cooler either contains a series of fans (active) or no fans (passive), the latter relying entirely on the fans in the servers to push the air through the back door cooler.

2. CFD modelling of thermal air flow in data centres

2.1. Mathematical model

Although Computational Fluid Dynamics (CFD) is now used to analyse and design thermal air flows in data centres, they are still largely unverified with respect to the accuracy of their thermal

* Corresponding author. Tel.: +44(0) 113 343 2136; fax: +44(0)113 343 2150.
E-mail address: h.m.thompson@leeds.ac.uk (H. Thompson).

0306-2619/$ - see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.apenergy.2011.02.003

predictions for large data centres [6]. In addition, the modelling methodologies adopted for the all-important flow through server racks have not been described adequately [1], making comparison with previous studies extremely difficult. In this paper we briefly describe a modelling methodology for the thermal air flow in data centres with a particular focus on the representation of the server racks and back door cooling units.

Thermal air flows in data centres are usually complex, recirculating air flows characterised by a hierarchy of different length scales. A typical Reynolds number, Re, based on an air inlet velocity from the supply vents of 1 m/s and a rack length scale of 2.4 m, leads to an estimated $Re \approx 10^5$ indicating the turbulent flow regime. Most previous CFD studies of data centre air flows have used Reynolds Averaged Navier–Stokes (RANS) models, see e.g. the very recent study of Cho et al. [1]. The governing continuity and momentum equations, written in the RANS format, are:

$$\underline{\nabla} \cdot \underline{U} = 0 \tag{1}$$

$$\frac{\partial \underline{U}}{\partial t} + \underline{\nabla} \cdot (\underline{U}\underline{U}) = \frac{1}{\rho}\underline{\nabla} \cdot (\underline{\underline{\sigma}} - \rho\overline{\underline{U}'\underline{U}'}) + \frac{1}{\rho}\underline{S} \tag{2}$$

where $\underline{\underline{\sigma}} = -P\underline{\underline{I}} + \mu(\underline{\nabla}(\underline{U}) + [\underline{\nabla}(\underline{U})]^T)$ is the Newtonian stress tensor and μ is the air viscosity, ρ its density (defined using the ideal gas law with constant pressure 1.013×10^5 Pa), $\underline{U}$ and $\underline{U}'$ are the average and turbulent fluctuation velocity vectors respectively, P is the pressure and $\underline{\underline{I}}$ is the unit tensor. The vector $\underline{S}$ represents the additional momentum sources, which will be discussed in greater detail below, and the $-\rho\overline{\underline{U}'\underline{U}'}$ term is the so-called Reynolds stress tensor that requires additional model equations.

Following Cho et al. [1], the CFD models developed here use the standard k–ε model where the turbulence is described with two additional variables k (turbulent kinetic energy) and ε (turbulent dissipation). This model requires the flow to be fully turbulent, which may not be the case in all areas of the air flow within the data centre, but the model is well tested and popular for flows that have complex geometries and heat transfer. Clearly, more detailed investigations into the most appropriate turbulence modelling approaches are required, however this issue is not pursued here. The two transport equations are:

$$\frac{\partial k}{\partial t} + \underline{\nabla} \cdot (k\overline{U}) = \frac{1}{\rho}\underline{\nabla} \cdot \left(\frac{\mu_t}{\rho_k}\underline{\nabla}(k)\right) + \frac{2\mu_t}{\rho}S_{ij} \cdot S_{ij} - \varepsilon \tag{3}$$

$$\frac{\partial \varepsilon}{\partial t} + \underline{\nabla} \cdot (\varepsilon\overline{U}) = \frac{1}{\rho}\underline{\nabla} \cdot \left(\frac{\mu_t}{\rho_\varepsilon}\underline{\nabla}(\varepsilon)\right) + C_{1\varepsilon}\frac{\varepsilon}{k\rho}2\mu_t S_{ij} \cdot S_{ij} - C_{2\varepsilon}\frac{\varepsilon^2}{k} \tag{4}$$

with the turbulent viscosity defined via

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{5}$$

and the S_{ij} terms are the deformation tensor. Five empirical constants ρ_k, ρ_ε, $C_{1\varepsilon}$, $C_{2\varepsilon}$, and C_μ in Eqs. (3)–(5) are set equal to 1, 1.3, 1.44, 1.92, and 0.09 respectively (Boulet et al. [7]).

The energy equation is also solved and takes the form:

$$\frac{\partial T}{\partial t} + \underline{\nabla} \cdot (T\underline{U}) = \underline{\nabla} \cdot \left(\left(\frac{\upsilon}{Pr} + \frac{\upsilon_T}{Pr_T}\right)\underline{\nabla}(T)\right) + \frac{1}{\rho C_p}S_Q \tag{6}$$

where T and υ are the temperature and dynamic viscosity respectively and Pr is the Prandtl number defined by

$$Pr = \frac{\upsilon}{\alpha} \quad \text{where} \quad \alpha = \frac{k}{\rho C_p}, \tag{7}$$

k is the thermal conductivity and C_p is the air's specific heat transfer capacity. The subscript T indicates the turbulent flow and S_Q is the source term of the energy equation, described below.

2.2. Server rack and back door cooler modelling

The simplified data centre configuration considered is shown in Fig. 1. It consists of two rows of three server racks where cold air is supplied from six floor vents into a cold aisle, passes the rack mounted IT equipment (servers), absorbs the heat generated by

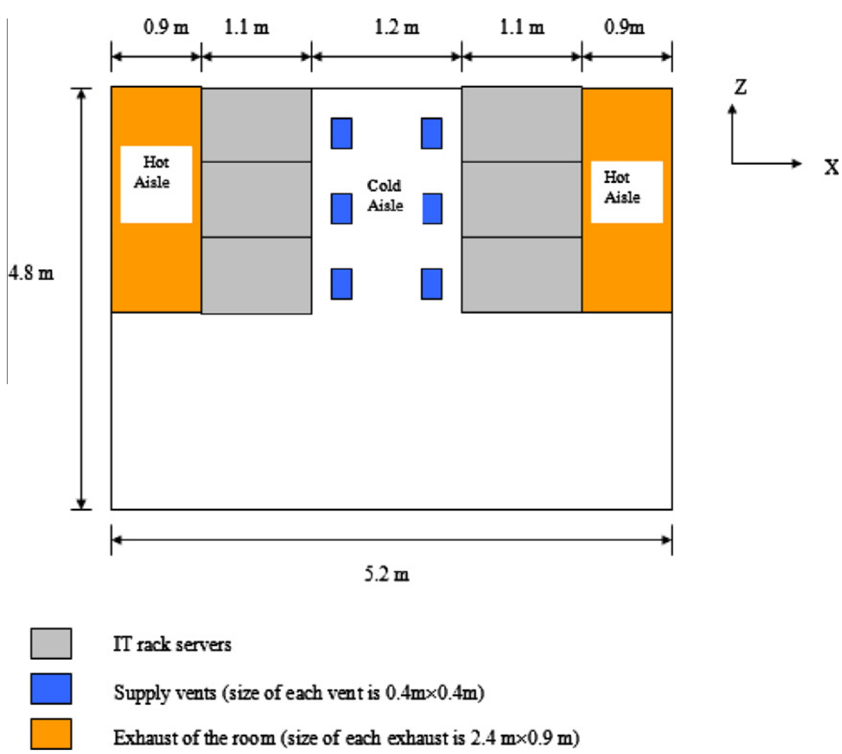

Fig. 1. Plan view of three-aisle, six-rack data centre.