Available online at www.sciencedirect.com
ScienceDirect
Applied Thermal Engineering 28 (2008) 1206–1215
APPLIED THERMAL ENGINEERING
www.elsevier.com/locate/apthermeng

Evaluation of thermal comfort in Galatsi Arena of the Olympics "Athens 2004" using a CFD model

Anastasios I. Stamou [a,*], Ioannis Katsiris [a], Alois Schaelin [b]

[a] School of Civil Engineering, National Technical University of Athens, Iroon Polytechniou 5, 15780 Athens, Greece
[b] Air Flow Consulting, Weinbergstrasse 72, CH-8006 Zürich, Switzerland

Received 24 May 2007; accepted 31 July 2007
Available online 24 August 2007

Abstract

A computational fluid dynamics (CFD) model was used to evaluate the thermal comfort conditions in the indoor stadium of the Galatsi Arena, which hosted the sports of rhythmic gymnastics and table tennis during the Olympic Games "Athens 2004". The CFD code CFX was applied to calculate the 3D airflow and temperature fields in the Arena for various values of temperatures (T_{in}) of conditioned inlet air. Calculated mean velocities and temperatures were used to determine the thermal comfort indices predicted mean vote (PMV) and predicted percentage of dissatisfied (PPD) and to evaluate the thermal conditions in the various regions of the Arena. Calculated PMV and PPD values showed that thermal conditions in the Galatsi Arena were very satisfactory for $T_{in} = 16$ °C; only a small percentage (<7%) of the spectators was expected to be slightly uncomfortable.

Keywords: Computational fluid dynamics (CFD); Indoor stadiums; Mathematical models; Thermal comfort

1. Introduction

In the last two decades computational fluid dynamics (CFD) models have been increasingly used for the calculation of airflow velocities and temperatures in indoor environments for the evaluation of comfort conditions, smoke conditions and air quality. A significant number or scientific papers exists dealing with the application of CFD models in various indoor environments, such as apartments [1], offices [2,3], museums [4], lecture theatres [5], classrooms [6], clinics [7], industrial premises [8] and car parks [9]. Moreover, CFD models have been combined with other tools or methods for the design of indoor spaces. For example, Desta et al. [10] combined a CFD model with a low order data based mechanistic (DBM) model to predict air temperatures in a full scale, adiabatic walled ventilated test chamber. The CFD model supplied data and the DBM model performed model order reduction; in this way the two modeling approaches supplemented each other symbiotically to eliminate their disadvantages. Zhai and Chen [11] analyzed the potential building and environmental characteristics (such as environmental conditions, HVAC systems, building occupying and operating conditions, envelope properties and building sizes) that may influence the necessity and effectiveness of applying energy simulation–CFD coupling simulation. They conducted a sensitivity analysis of the coupling simulation in a representative office building in Boston and provided general suggestions on appropriate development and usage of the coupling simulation. Kim et al. [12] presented a two-step optimal design investigation method using Genetic Algorithms for CFD indoor thermal environments and applied the optimal design in an office space.

In spite of the extensive application of CFD models in various indoor environments, there are only a few studies in indoor stadia. In 2001, a CFD model was applied for

* Corresponding author. Tel.: +30 210 772 2809; fax: +30 210 772 2814.
E-mail addresses: stamou@central.ntua.gr (A.I. Stamou), ykats@mail.ntua.gr (I. Katsiris).
URL: www.afc.ch (A. Schaelin).

1359-4311/$ - see front matter © 2007 Elsevier Ltd. All rights reserved.
doi:10.1016/j.applthermaleng.2007.07.020

Fig. 1. View of the Galatsi Arena (www.stadia.gr).

the Yoyogi National Stadium in Tokyo [13]. The Stadium was constructed for the Olympics of 1964. Since then, it has continued to be used for the performance of swimming competitions or concerts; the CFD model was used to evaluate indoor conditions for both modes of stadium operation. Very recently, a CFD study was performed in the Olympic ice hockey stadium in Turin to evaluate fire hazards; smoke conditions were reviewed in the stadium for various scenarios and uses [14].

The present work demonstrates the application of a CFD model to calculate air-flow velocities and temperatures in the Galatsi Arena, shown in Fig. 1; the Arena hosted the sports of rhythmic gymnastics and table tennis during the Olympics of 2004. The calculated velocities and temperatures were used to determine the main thermal comfort indices predicted mean vote (PMV) and predicted percentage of dissatisfied (PPD) [15]; these were subsequently used for the evaluation of the thermal conditions in the Arena.

2. The mathematical model

2.1. The CFD code

There are three main types of CFD methods: (1) direct numerical simulation (DNS), (2) large eddy simulation (LES) and (3) Reynolds averaged Navier–Stokes (RANS). A short presentation of these methods can be found in Stamou and Katsiris [3]. These methods are employed in various efficient computer codes, which are most frequently used for indoor CFD calculations, such as PHOENICS [16], FLUENT [17] and CFX [18]. In the present work, the latest version of the computer code CFX [18] was used.

The CFX code calculates the 3D flow field and heat transfer using the continuity, momentum and energy equations, which are written as follows:

$$\frac{\partial \rho U_j}{\partial x_j} = 0 \tag{1}$$

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial \rho U_j U_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu \frac{\partial U_i}{\partial x_j} + \mathrm{RTS}\right) + g_i \rho \tag{2}$$

$$\frac{\partial T}{\partial t} + \frac{\partial \rho U_j T}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\mu \frac{\partial T}{\partial x_j} + \mathrm{TTS}\right) \tag{3}$$

where t is the time, x_i is the Cartesian coordinate in the i-direction, U_i is the flow velocity in the i-direction, ρ is the density of air, T is the temperature, P is the pressure, μ is the molecular viscosity of the air and g_i is the acceleration of gravity.

For the calculation of the Reynolds (turbulent) stresses (RTS), the assumption of the isotropic turbulence is applied combined with the Boussinesq approximation, i.e.

$$\mathrm{RTS} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \frac{2}{3}\rho k \delta_{ij} \tag{4}$$

where μ_t is the eddy viscosity, δ_{ij} is the Kronecker delta ($\delta_{ij} = 1$ for $i = j$ and $\delta_{ij} = 0$ for $i \neq j$) and k is the average turbulent kinetic energy per unit mass, given by

$$k = \frac{1}{2}(\overline{u_1^2} + \overline{u_2^2} + \overline{u_3^2}) \tag{5}$$

Similarly, the turbulent thermal stresses (TTS) are calculated by the following equation:

$$\mathrm{TTS} = \frac{\mu_t}{\sigma_\mathrm{T}} \frac{\partial T}{\partial x_j} \tag{6}$$

where σ_T is the turbulent Schmidt number for T.

2.2. The turbulence model

In the present work, turbulence is modeled with the shear stress transport (SST) k–ω based model [19]. This model combines the standard k–ε model [20] and the k–ω model [21]; virtually, it is a transformation of the k–ε to