Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 126 (201709) 115–122

Energy Procedia

www.elsevier.com/locate/procedia

72nd Conference of the Italian Thermal Machines Engineering Association, ATI2017, 6-8 September 2017, Lecce, Italy

Prediction Of Indoor Conditions And Thermal Comfort Using CFD Simulations: A Case Study Based On Experimental Data

Cinzia Buratti[a], Domenico Palladino[a], Elisa Moretti[a]*

[a]*Department of Engineering, University of Perugia, Via G. Duranti 93, 06125 Perugia (PG), Italy*

Abstract

In the present paper CFD tool was used for thermal comfort evaluation in natural convection and in transient conditions in a room by setting only the external weather conditions as input parameters. A survey in a classroom at the Department of Engineering, University of Perugia, was carried out and data required for the thermal comfort evaluation and CFD simulation model set up was acquired. The simulation model was validated with experimental data and it was used for the thermal and velocity profiles simulation and for the thermal comfort indexes calculation, according to UNI 7730.

Peer-review under responsibility of the scientific committee of the 72nd Conference of the Italian Thermal Machines Engineering Association

Keywords: computational fluid dynamics (CFD); thermal comfort; building simulation; model validation; experimental data;

1. Introduction

In the modern society an increasing number of people spends most of their time in confined environments, with artificial climatic conditions, in which thermal comfort is a basic factor: glazing systems, both for dimensions and material characteristics, are very important because they influence the parameters involved in thermal comfort evaluation. Many studies were carried out in the recent years about thermal comfort in moderate environments by applying different kind of methods such as the classic approach introduced by Fanger [1] and the adaptive approach [2, 3]; the classic approach was introduced by Fanger [1], by means of the indexes Predicted Mean Vote (PMV) and Predicted Percentage of Dissatisfied (PPD), also adopted in EN ISO 7730 [4], which provides a method to calculate

* Corresponding author. Tel.: +39- 075-5853694; fax: +39-075-5853697.
E-mail address: elisa.moretti@unipg.it

Peer-review under responsibility of the scientific committee of the 72nd Conference of the Italian Thermal Machines Engineering Association
10.1016/j.egypro.2017.08.130

and to interpret global and local thermal comfort. However, the acquisition of the data necessary for the calculations requires specific instruments, not always available, and much time. Many studies are focused on the implementation of alternative tools for thermal comfort prediction; one interesting method could be the simulation implemented with CFD codes, which allow to simulate the thermal and the velocity profiles within environments [5, 6].

A wide bibliographic research was carried out in order to evaluate the state of the art of CFD applications in building thermal comfort predictions; several studies [5-15] were conducted in order to determinate the thermal comfort with CFD simulation in various indoor environments such us stadium [7], theatre [10], museum [12] or in a test room [13]. Stamou et al. [7] evaluated thermal comfort in a Galatsi Arena stadium with CFD simulations, considering heating, ventilating and air conditioning systems and assuming two possible inlet air temperatures: 14°C and 16°C. The calculated values of PMV and PPD showed that the thermal conditions were satisfactory when the inlet air temperature was equal to 16°C. Cheong et al. [10] evaluated the thermal conditions of an air-conditioned lecture theatre, both with experimental campaign and CFD simulations. It was shown that the values of temperature, air velocity and relative humidity were within the limits of thermal comfort standards. Papakonstantinou et al. [12] studied the velocity and temperature field in three dimensional simple geometry and in a museum, by setting external meteorology conditions, but without validating the model by means of experimental data. Catalina et al. [13] used a CFD model to evaluate the average velocity and temperature and the values of PMV in a test room with chilled ceiling panels; the results were first validated with experimental data and then the velocity fields were investigated.

In the present paper the application of a 3D CFD model simulation was used to support the experimental investigations, the temperature fields and the global and local thermal comfort sensation in a non-residential environment by setting only the external weather conditions, considering natural convection and the solar radiation influence [14-17].

2. Methodology

2.1. Experimental Campaign

An experimental campaign in a classroom at the Department of Engineering (University of Perugia) was carried out in the month of April when the HVAC system was turned off. All the parameters necessary for determining thermal comfort according to UNI 7730 [4] and for setting and validating the CFD model were measured. Specifically, the indoor and outdoor air temperatures, the relative humidity of air, the indoor air speed, the globethermometer temperature, the air pressure, the internal and external solar radiation on a vertical plane, the surface temperatures of opaque and transparent walls were acquired. The heat flux through the opaque wall was also monitored in order to calculate the equivalent thermal conductivity of the opaque wall and it was set as input data on the CFD model. The technical features of the measurement equipment are already described in [18] with uncertainty of measurement of about ±2÷7%.

In addition to the thermal characteristics of the external walls, only the outdoor air temperature and the solar radiation were set as input parameters, while the temperatures of opaque and transparent surfaces and the indoor air temperature monitored within the classroom were chosen for validating the simulation model as showed in previous papers [5-15]. The solar radiation was set up by using the solar model available in the CFD code. In Fig. 1 the geometrical characteristics of the external wall and the plant of the classroom are reported: the position of the measurement points is highlighted and a view of the classroom during the experimental campaign is shown.

2.2. CFD model and preliminary settings

A CFD solver package, ANSYS Fluent, was used to perform all the CFD computations; it allows to evaluate the thermal and flow fields based on continuity, momentum, and heat transfer equations already described in [7]; in particular, in agreement with a previous work [5], the energy model and the k-ε model were used. In addition, two additional equations were implemented: the Boussinesq approximation, which allows to simulate the natural convection, and the solar model for the solar gain [19, 20]. The transient condition was simulated by setting a specific simulation time size in the CFD code; this value also depends on the implemented equations. Considering