Case Studies in Thermal Engineering 4 (2014) 187–192

Contents lists available at ScienceDirect

Case Studies in Thermal Engineering

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

Temperature optimization of an electric heater by emissivity variation of heating elements

Cédric Hemmer [a,b], Guillaume Polidori [a], Catalin Popa [a,*]

[a] GRESPI/Thermomécanique, UFR Sciences Exactes et Naturelles, Campus du Moulin de la Housse, BP 1039, 51687 Reims, France
[b] CAMPA, rte Soissons, 51170 Fismes, France

ARTICLE INFO

Article history:
Received 23 June 2014
Received in revised form
7 October 2014
Accepted 8 October 2014
Available online 18 October 2014

Keywords:
Natural convection
Radiation
Electric heater
Emissivity
CFD

ABSTRACT

This note addresses an industrial application concerning the way to optimize the surface temperature of commercial electrical heater. The aim of this paper is to reduce the temperature on accessible surfaces and electrical heater in order to respect the European standards and quality criteria imposed by the manufacturer. This target must be achieved by changing only the emissivity distribution of the electric heater components. A numerical study of the natural convection flow coupled with radiation is carried out in a heated room with an electric heater. The physical model includes the transport equations of mass, momentum, energy and radiative transfer which are solved numerically. Thermo-physical properties of the fluid are assumed to be dependent of the temperature. The numerical simulations are carried out for a two-dimensional, steady and turbulent flow using the finite volume approach. Results showed the influence of emissivity distribution of the electric heater components. The reducing of the heating foil emissivity allowed to decrease the radiative contribution on the foil and its temperature.

1. Introduction

Thermal comfort in buildings is an important property for the quality of indoor environments, but also for the amount of energy required by the equipment's room. In addition, the energy consumption of systems is a highly watched topic in current thermal regulations. It is therefore important to understand and control the exchange of natural convection airflow coupled with radiation coming into play in domestic heating units.

Several models have been developed for a better understanding and prediction patterns of heat transfer in buildings. Some are traditional models based on energy simulations (zonal models) and others based on the resolution of conservation equations (Computational Fluid Dynamics models). The zonal models based on energy simulation models are monozone or multizone. The principle is based on the division of the room volume in a limited number of zones at a constant airflow behavior, such as the plume area on top of radiator or the area in contact with the floor. Very encouraging results have been obtained by Inard et al. [1]. The authors were able to show that the zonal method can give good predictions in the temperature distribution in different scenarios (electric heater, water heater and floor heating). However, Bezzo et al. [2] note that the zonal method poses great difficulties to characterize both the flow between adjacent zones, and certain mechanical quantities such as the dissipation rate of turbulent energy, which have important effects on the existing process in each area. These types of models are very useful for simulating the behavior of an annual installation and determine its

* Corresponding author.
E-mail address: catalin.popa@univ-reims.fr (C. Popa).

http://dx.doi.org/10.1016/j.csite.2014.10.001
2214-157X/

Nomenclature

Symbols

a	absorption coefficient
C_p	specific heat capacity, J/kg K
g	acceleration of gravity, m/s^2
k	thermal conductivity, W/m K
H	room height, m
I	radiation intensity
L	room width, m
n	refractive index
P	pressure, Pa
P^*	driving pressure, Pa
$\vec{r}$	position vector
s	path length
$\vec{s}$	direction vector
$\vec{s}\,'$	scattering direction vector
T	temperature, K
u,v	axial and transverse velocity components, m/s
x,y	axial and transverse coordinates

Greek letters

ε	emissivity
ρ	density, kg/m^3
ν	kinematic viscosity, m^2/s
μ	dynamic viscosity, Pa s
σ	scattering coefficient
ς	Stefan–Boltzmann constant
Ω	solid angle
Φ	phase function

consumption. However, this type of model does not have a very high precision in the spatial distribution of temperature and velocity fields in a room. Mora et al. [3] compared the zonal method and the Computational Fluid Dynamics (CFD) method. They observed an increase of the accuracy for CFD models even if the zonal model appears to give good results. Models based on solving the conservation equations of mass, momentum and energy expensive in computation time, allow us to obtain better accuracy. Several studies of CFD applied to the field of the building have already been made. These studies have shown the feasibility of the use of CFD codes to predict the thermo-aeraulic behavior on the room scale [4,5]. In these two articles, one may note that the CFD method is feasible and very efficient to predict the velocity and temperature fields in a room. However, a large number of these items mainly studied the influence of ventilation or air conditioning system in forced convection and often without taking into account the radiation [6–8]. Sevilgen et al. [9] conducted a study in natural convection taking into account the radiation with a simplified model of heater. Moreover, Sharma et al. [10] and Borjini et al. [11] showed that the radiation has an important role and can change behavior of the thermo-aeraulic field in natural convection flow.

This paper presents a numerical study of the natural convection flow coupled with radiation in a heated room with an electric heater. The dynamic and thermal flows studied within and in close proximity to an electric heater are turbulent and steady. Two-dimensional numerical simulations have been carried out with the finite volume approach. In this study, a manufacturer of electric heater wants to reduce the temperature of some of the elements constituting the heater without changing the powers and geometries of these heaters. The investigations focus specifically on the influence of emissivity distribution of the electric heater components to decrease the temperature on accessible surfaces and electrical heater in order to respect the European standards [12] and quality criteria imposed by the manufacturer.

2. Problem setup

2.1. Geometry

The electric heater is placed in a rectangular room where the height is H=2.6 m and length L=3.6 m (Fig. 1).

The electric heater used for the numerical simulations has three heating elements. The first element, a heating foil ① is placed on the front to obtain, on the front face a surface with a uniform heat flux density. The heating foil has a power of 400 W with a height of 0.484 m, a width of 0.730 m and a thickness of 0.001 m. The facade is composed of the heating foil adhered to an opaque infrared glass plate ② with an 8 mm thickness. The other two heating elements are aluminum electrical resistances ③ joined together, each 300 W power. This electrical power ⑤ of 300 W is applied only to the inner cylinder. The box ④ of the electric heater is painted. Electric heater has a total nominal power of 1000 W and a height of 565 mm, a width of 820 mm and a thickness of 121 mm.

2.2. Physical model

The physical model of natural convection flow coupled with radiation in a cavity includes the transport equations of mass, momentum, energy [13] and radiative transfer [14] which are solved numerically using the finite volume method [15]. The numerical simulations are carried out for a two-dimensional, steady and turbulent flow. Thermo-physical properties of the fluid are assumed to be dependent of the temperature because Boussinesq approximation is not valid in used temperature range [16]. The radiative transfer equation [14] for an absorbing and scattering medium emissivity at a position