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

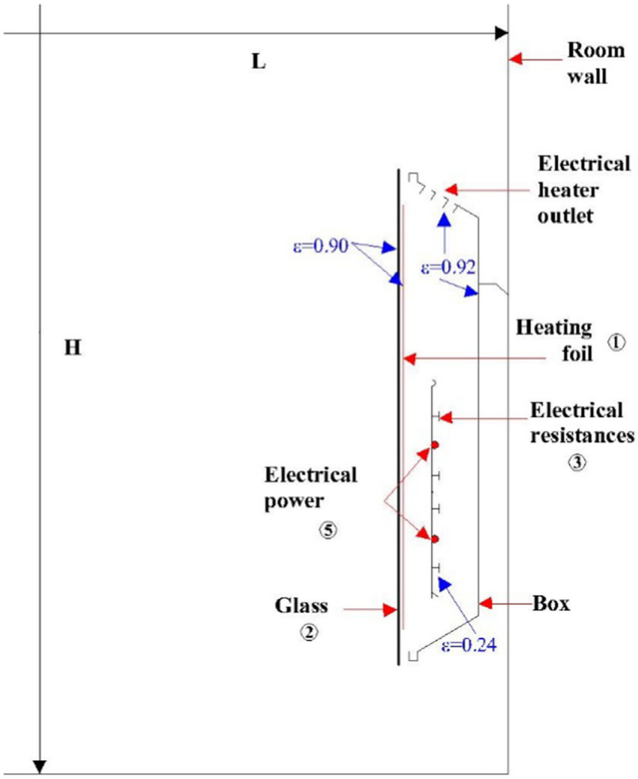

Fig. 1. Synopsis of the device.

$\vec{r}$ in the direction $\vec{s}$ is

$$\frac{dI(\vec{r},\vec{s})}{ds}+(a+\sigma)I(\vec{r},\vec{s})=an^2\frac{\varsigma T^4}{\pi}+\frac{\sigma}{4\pi}\int_0^{4\pi}I(\vec{r},\vec{s}\,')\,\Phi(\vec{s}.\vec{s}\,')d\Omega \tag{1}$$

Boundary conditions are as follows:

- At the room wall: $T=15\ ^{\circ}\mathrm{C}$
- At the heating foil: $P=400$ W
- At the electrical resistances: $P=300$ W

The emissivities quoted are measured on the product currently manufactured. These emissivities are due to materials used without specific surface treatment. The elements emissivities for the initial case are as follows:

- At the room wall: $\varepsilon=0.95$
- At the glass: $\varepsilon=0.9$
- At the heating foil: $\varepsilon=0.9$
- At the electrical resistances: $\varepsilon=0.24$
- At the box: $\varepsilon=0.92$

2.3. Numerical method

Transport equations of mass, momentum and energy are solved numerically using finite volume method [15]. This method is based on the spatial integration of transport equations relative to control volumes. The coupling between velocity and pressure is achieved with the algorithm "coupled scheme" that solves the equations of continuity and momentum simultaneously and gives an advantage to treat flows with a strong interdependence between dynamic and thermal fields. Also, in order to model the radiation one may use the radiation model "Surface to Surface" with a method of form factors. This model takes into account the radiative heat exchange between gray and Lambertian walls [17]. The natural convection flow coupled with radiation in a cavity is turbulent. Turbulence is modeled with the k–ω SST turbulence model [18,19]. Numerical simulations are performed with ANSYS Fluent® CFD commercial software. The mesh inside electrical heater was made fine and structured. The mesh (89,000 cells) is structured and refined near the wall to satisfy the turbulence model condition($y^+ = 1$). This mesh becomes coarser away from the wall in order to optimize the computational time. The convergence criteria were based on the residuals resulting from the integration of the conservation equations over finite control volumes. During the iterative calculation process, these residuals were constantly monitored and carefully scrutinized. For all simulations performed in this study, converged solutions were achieved with residuals as low as 10^{-5} (or less) for all the governing equations.

3. Model validation

Before beginning any studies on the thermal behavior of the flows obtained by the numerical simulation, it is necessary to check that the model developed is reliable. To perform this step, numerical results are compared with experimental data. A first simulation is performed with a total power of 1000 W for the electrical heater. In this simulation, we use the boundary conditions detailed in the physical model.

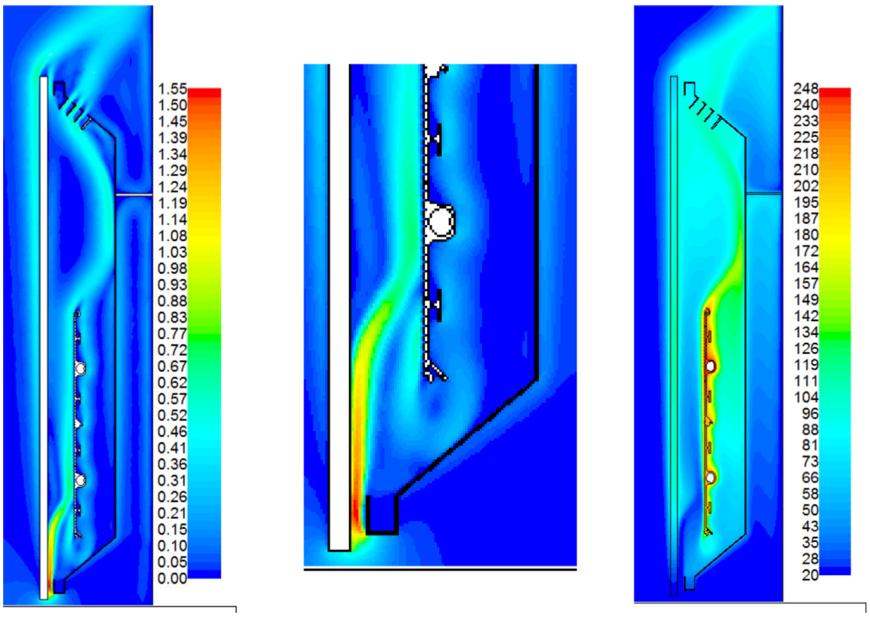

Fig. 2. (a) Velocity field in the electrical heater. (b) Velocity field in the entry of electrical heater. (c) Temperature field in the electrical heater.

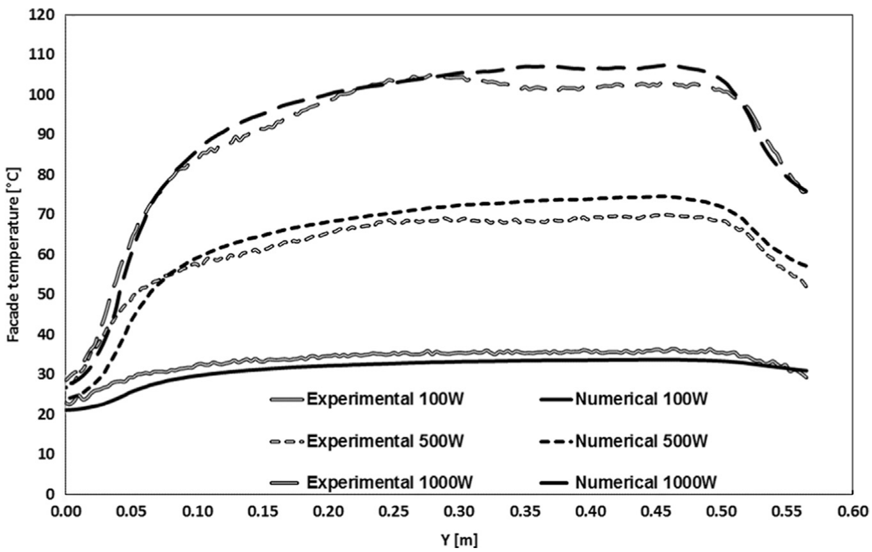

Fig. 3. Comparison between experimental and numerical façade temperature profiles for several heating power.

In Fig. 2a–c one may observe the velocity magnitude and temperature in the electrical heater. An ascending boundary-layer flow develops near the heated foil and electrical resistances due to a fluid supply from the bottom of the electrical heater. One may observe the development of a dynamic boundary layer in the lower part of the radiator along the heating foil ①. After 8.5 cm from the electrical heater entry, a separation phenomenon of the dynamic boundary layer occurs. This is due to the higher temperature of the electrical resistances ③ which induces a fluid aspiration from the left part to the right one (Fig. 2b). On the glass surface a development of dynamic boundary layer is also observed. The electrical resistances are the warmer radiator elements whose temperature is 243.5 °C. One can notice that the temperature of the heating film, composed by polyethylene, is lower: 116 °C. This temperature is nevertheless close to the melting point of polyethylene (120 °C).

Validation by thermography was executed on the front façade of the electrical heater to the initial case with several heating power (100 W, 500 W and 1000 W). The electrical heater has been tested in accordance with the same boundary conditions. Temperature measurement uncertainty with the method of infrared thermography is 2 °C.

In Fig. 3 the façade temperature profiles have been plotted versus the electrical heater height. One may observe the similar trends between experimental and numerical temperature profile. The maximum relative difference observed between the experimental and numerical values was approximately 7%.

4. Results and discussion

In order to obtain a more secure and more sustainable product, the manufacturer must reduce the temperature of the heating elements. To perform this purpose, the manufacturer needs to change the temperature distribution in electrical heater by changing the emissivity of the heating elements.

The thermal field in the natural convection flow coupled with radiation has been investigated numerically. In this study the emissivity of the heating elements was modified (Table 1) to evaluate its influence on the maximal temperature of accessible surfaces and electrical heater outlet. The change of the electrical resistance emissivity (cases 1 and 3) can be explained by the surface treatment that the fabricant can apply it. This surface treatment consists in painting the raw aluminum resistances (emissivity $\varepsilon=0.24$) with a specific paint (emissivity $\varepsilon=0.98$). The heating foil initially has an emissivity of $\varepsilon=0.90$. The change of the heating foil emissivity (cases 2 and 3) is due to the application of an aluminum reflective layer ($\varepsilon = 0.24$). These changes of specific emissivity have been chosen to their industrial simplicity and low cost of implementation. These changes should not degrade the performance of the device or should not make it more dangerous for the user. Indeed, each element has its own critical temperature. The heating foil is made of plastic and should not exceed 120 °C to prevent the fusion. The accessible parts of the user, namely the facade, the box and the air outlet must not exceed a maximum temperature of 110 °C to prevent accidental burn [12]. Also, so as not dry out the ambient air, the electrical resistances must not exceed a maximum temperature of 250 °C.

In order to reduce the temperature of the heating foil and thus avoid its destruction, one may test different emissivity settings (Table 1) for heating elements.

In the initial case, the heating foil is near its maximum temperature which could cause premature usury of the heating foil. Also the temperature facade is also very close to the maximum temperature. It is therefore necessary to reduce temperature of these components in order to have a more efficient device, in respect with reliability and user safety. For the other elements constituting the heater, one may note that the temperature are within the limits specified.

Table 1
Distribution of the heating elements emissivity.

Emissivity (ε)	Heating foil	Electrical resistance
Initial case	0.9	0.24
Case 1	0.9	0.98
Case 2	0.24	0.24
Case 3	0.24	0.98

Table 2
Maximal temperature for several testing cases.

Maximal temperature (°C)	Initial case	Case 1	Case 2	Case 3
Heating foil	116.0	129.4	105.9	108.9
Facade	107.2	118.9	98.6	101.0
Electrical resistances	243.5	198.0	247.6	212.1
Box	97.5	107.8	97.9	117.5
Air outlet	105.0	94.3	105.7	94.8

For case 1, the emissivity increase of the electrical resistances has the effect of reducing the temperature of these components. However, the maximum temperatures of heating foil and facade are exceeded. Furthermore, one may observe an increasing of the box temperature and consequently behind the heater. This configuration is not relevant.

With case 2, one may note a significant decrease in the temperature of the heating foil, and of the facade compared to the initial case. This decrease is accompanied by a very slight increase in the internal temperature of the electrical resistances. Other elements of the heater are at substantially the same temperature as the initial case. This configuration is very interesting because it leads to lower temperatures on the most critical elements.

In case 3, one may observe that the façade, the heating foil and electrical resistances temperature decreases compared to the initial case but the box temperature increases. Finally, to produce a heater, energy efficient, it is necessary that the product supplied energy to the user located in front, rather than on the back wall. Thereby, a box hotter than a façade is not a feasible solution for the manufacturer. According to the values obtained in Table 2, case 2 is the only case to satisfy all criteria defined previously. In this case, the radiative contribution of the hotter electric resistances on this heating foil decreases by reducing the emissivity of the heating foil.

5. Conclusion

This work has studied the influence of emissivity variation scenarios of different electric heater components to reduce the temperature on accessible surfaces and electrical heater outlet in order to respect the European standards [12] for an industrial application. For this purpose, a numerical study of the natural convection flow coupled with radiation in a heated room with an electric heater has been led. The dynamic and thermal flows have been studied within and in close proximity to an electric heater. Two-dimensional numerical simulations have been carried out with the finite volume approach. Results were presented in the steady state for the turbulent flow. The numerical model of natural convection with radiation used in this study was validated with experimental data. Results showed the influence of emissivity distribution of the electric heater components. One may notice in case 2 a temperature increase of 2% of the electrical resistance. However, this case allows us to reduce the temperature of the heating foil by 9%. In this case, the radiative contribution of the hotter electric resistances on this heating foil decreases by reducing the emissivity of the heating foil. In conclusion, all the heating elements that make up an electric heating device must have low emissivity.

Acknowledgments

This work is financially supported by the European Union (EU) through the European Regional Development Fund (FEDER), the Champagne-Ardenne region and the company CAMPA as part of Effi-SiEMCE project.

References

[1] Inard C, Bouia H, Dalicieux P. Prediction of air temperature distribution in buildings with a zonal model. Energ Build 1995;24:125–32.
[2] Bezzo F, Macchietto S, Pantelides CC. A general methodology for hybrid multizonal and CFD models: Part I theoretical framework. Comput Chem Eng 2003:501–11.
[3] Mora L, Gadgil AJ, Wurtz E, Inard C. Comparing zonal and CFD model predictions of indoor airflows under mixed convection conditions to experimental data. In: Proceedings of the EPIC'02 conference; 2002. p. 735–40.
[4] Lu W, Howarth AT, Jeary AP. Prediction of airflow and temperature field in a room with convective heat source. Build Environ 1997;32:541–50.
[5] Fengying Y, Huaigang L, Long W, Haixia Z. Simulation of indoor dynamic thermal comfort based on CFD. In: Proceedings of the Fourth International Conference on Intelligent Computation Technology and Automation; 2011.
[6] Hanqing W, Chunhua H, Zhiqiang L, Guangfa T, Yingyun L, Zhiyong W. Dynamic evaluation of thermal comfort environment of air-conditioned buildings. Build Environ, 41(11):1522–9.
[7] Chen, Qingyan. Ventilation performance prediction for buildings: a method overview and recent applications. Build Environ 2009:848–58.
[8] Sun Z, Wang S. A CFD-based test method for control of indoor environment and space ventilation. Build Environ 2010;45:1441–7.
[9] Sevilgen G, Kilic M. Numerical analysis of air flow, heat transfer, moisture transport and thermal comfort in a room heated by two-panel radiators. Energy Build 2011:137–46.
[10] Sharma AK, Velusamy K, Balaji C, Venkateshan SP. Conjugate turbulent natural convection with surface radiation in air filled rectangular enclosures. Int J Heat Mass Transf 2007;50:625–39.
[11] Borjini MN, Ben Aissia H, Halouani K, Zeghmati B. Effect of radiative heat transfer on the three-dimensional Boyancy flow in cubic enclosure heated from the side. Int J Heat Fluid Flow 2008;29:107–18.
[12] International Electrotechnical Commission (IEC) 60675. Household electric direct-acting room-heaters. Methods for Measuring Performance; 1998.
[13] Bejan A. Convection heat transfer. Hoboken (N.J.): John Wiley & Sons; 2004.
[14] Sparrow EM. Radiation heat transfer. Belmont, California: Brooks Pub. Co.; 1966.
[15] Patankar SV. Numerical heat transfer and fluid flow. Hemisphere, New York: Hemisphere Publishing Corporation, Mc. Graw Hill Book Company; 1980.
[16] Gray DD, Giorgini A. The validity of the Boussinesq approximation for liquids and gases. Int J Heat Mass Transf 1975;19:545–51.
[17] Colomer G, Cònsul R, Oliva A. Coupled radiation and natural convection: different approaches of the SLW model for a non-gray gas mixture. J Quant Spectrosc Radiat Transf 2007;107:30–46.
[18] Zang Z, Zhang W, Zhai Z, Chen Q. Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part 2 comparison with experimental data from literature. HVAC&R Res 2007;13:871–86.
[19] Menter FR. Review of thee shear-stress transport turbulence model experience from an industrial perspective. Int J Comput Fluid Dyn 2009;24: 305–16.