Applied Thermal Engineering 94 (2016) 266–273

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research Paper

CFD modelling of radiators in buildings with user-defined wall functions

Daniel Risberg *, Mikael Risberg, Lars Westerlund

Energy Engineering, Division of Energy Science, Luleå University of Technology, SE-971 87 Luleå, Sweden

HIGHLIGHTS

- This paper studies how to deal with natural convective heat transfer for a radiator in order to simplify the simulations.
- By adding user-defined wall functions the number of cells can be reduced considerably compared with the k-ω SST turbulence model.
- Compared to manufacturer data the error of the model is less than 0.2% for the investigated radiator height and temperature.

ARTICLE INFO

Article history:
Received 6 July 2015
Accepted 23 October 2015
Available online 3 November 2015

Keywords:
CFD modelling
Radiator
User-defined wall functions
Indoor climate

ABSTRACT

The most widely used turbulence model for indoor CFD simulations, the k-ε model, has exhibited problems with treating natural convective heat transfer, while other turbulence models have shown to be too computationally demanding. This paper studies how to deal with natural convective heat transfer for a radiator in order to simplify the simulations, and reduce the numbers of cells and simulation time. By adding user-defined wall functions the number of cells can be reduced considerably compared with the k-ω SST turbulence model. The user-defined wall function proposed can also be used with a correction factor for different radiator types without the need to resolve the radiator surface in detail. Compared to manufacturer data the error is less than 0.2% for the investigated radiator height and temperature.

© 2015 Elsevier Ltd. All rights reserved.

1. Introduction

The indoor climate in buildings is often modelled using building simulation software that predicts the indoor climate as well as the mixed airflow with a uniform velocity in each room. This method does not meet the requirements for detailed indoor climate analysis where it is necessary to predict the air velocity and temperature at different locations inside the room [1].

With CFD simulation technique the indoor climate can be predicted in the whole simulation volume. Therefore the velocity and temperature distribution in a building can be studied in detail [2]. Several studies of simulation of the indoor climate have been carried out but usually focusing on single rooms or specific areas inside a building [3]. These limitations are mainly caused by the amount of cells in the computational mesh being too huge for simulation of an entire building. One way to be able to simulate the indoor climate in a complete building is to simplify the model to reduce the number of cells in the simulation volume.

For indoor climate simulations the most common turbulence model used is the k-ε model [4] with scalable wall treatment. Previous studies using this model have shown good agreement for predicting the airflow within the room [5] and prediction of forced convective flow. However, the model is not accurate for predictions of natural and mixed convective flows [6]. In buildings this phenomenon is the main driving force for the indoor airflow in most situations and is therefore important to predict correctly. The scalable wall treatment has been shown to underestimate the convective heat transfer coefficients [7]. An underestimation of the convective heat transfer coefficients will result in a large temperature difference between surfaces and the indoor air, which will result in an over-prediction of the radiation heat transfer from the surfaces. This phenomenon has a large effect on the temperature of the warmest and coldest surfaces inside the building, for example radiators and windows.

Other turbulence models have been used to simulate indoor climate with good agreement and in order to predict the mean velocity and temperature gradient where large eddy turbulence models show the most accurate results [8]. The main drawbacks of large eddy simulation are the long computational time and large memory requirements, and it is therefore not used for simulations of large volumes. The main reason for this is the treatment of the near wall region where a very fine computational mesh is required that gives y + values (dimensionless distance from the wall) around 1; furthermore the simulation needs to be time resolved. Also the k-ω

* Corresponding author. Tel.: +46 920 491662; fax: +46 911 491399.
E-mail address: daniel.risberg@ltu.se (D. Risberg).

http://dx.doi.org/10.1016/j.applthermaleng.2015.10.134
1359-4311/© 2015 Elsevier Ltd. All rights reserved.

SST model has shown good agreement with measurement for simulation of indoor airflow with heat transfer [9]. The k-ω SST model also needs y + values around 1 and will require a higher number of cells compared to the k-ε model, which allows higher y + values. Therefore the k-ε model can use a courser mesh and the computational time will decrease significantly compared to other models.

The aim of this work was to investigate if the k-ε model could be used as turbulence model in order to correctly describe the indoor climate for a room with radiators as heat distribution system. Another objective was to simplify the geometry of a radiator in the model to be able to decrease the number of cells in the simulation volume.

2. Theory

2.1. Radiator type

The most common type of radiator is the panel radiator consisting of one or several panels to distribute the heat to the room (see Fig. 1). Different kinds of radiators have a different amount of convective heat transfer, which depends on the numbers of panels and the area of the radiator. The amount of radiation and convection heat transfer for a typical panel radiator is presented in Table 1.

2.1.1. Radiator standard testing

A radiator in Europe is tested according to European Standard EN 442-2 [11]. The test is performed with a 1 metre wide radiator for each height. The supply water temperature to the radiator is specified at 75 °C, and the water mass flow into the radiator is adjusted to give a return temperature of 65 °C. During the experiment all surrounding surfaces in the room are uniformly cold except the wall behind the radiator, which is insulated. The cooling rate is adjusted so that an indoor air temperature of 20 °C is obtained at a distance of 1 m ahead of the radiator and 1 m above the floor. During the experiment the heat transfer rate from the radiator is determined. The dimensions of the testing room are (L × W × H) 4 × 4 × 3 m, and the radiator is placed at a distance of 0.05 m from the wall and at a height of 0.10 m above the floor level.

Fig. 1. Radiator with two panels [10].

Table 1
Amount of convection and radiation heat transfer from different types of radiators in an isothermal cooled surface enclosure [11].

Type	Radiation heat transfer [%]	Convection heat transfer [%]
Single panel	50	50
Double panels	30	70
Triple panels	25	75

In order to decide the heat transfer rate (Φ) for a radiator with other widths and temperatures, a formula called the one exponent formula is used [12].

$$\Phi = \Phi_n b / (\Delta T / \Delta T_n)^n. \tag{1}$$

where (Φ_n) stands for the determined heat transfer rate according to EN 442-2, and b is the width of the radiator in unit metre. The logarithmic temperature ΔT is the temperature difference between the air in the reference point and the radiator mean surface temperature; it is calculated according to Equation (2). The exponent n in Equation (1) is the radiator exponent that usually is in the range of 1.2–1.4 for all types of radiators. The most common value is around 1.3 [13]. ΔT_n is the logarithmic temperature for the normal test case (water temperature 75/65 °C) with an air room temperature of 20 °C.

$$\Delta T = \frac{T_{supply} - T_{return}}{\ln((T_{supply} - T_{ref})/(T_{return} - T_{ref}))} \tag{2}$$

where T_{supply} is the water supply temperature, T_{return} is the water return temperature from the radiator, and T_{ref} is the air temperature at the reference point.

2.2. Convective heat transfer calculations

The main problem for convective heat transfer is to determine the boundary conditions at a surface exposed to a flowing fluid. The local Rayleigh number (Ra_x) is calculated as

$$Ra_x = \frac{g\beta (T_w - T_{ref}) x^3}{\nu\alpha} \tag{3}$$

where g is the acceleration of gravity, β the thermal expansion coefficient, T_w is the wall (surface) temperature, x is the vertical position on the radiator surface, ν is the kinematic viscosity, and α is the thermal diffusivity. The Nusselt number (Nu_x) is established by [14] according to Equation (4). The Prandtl number (Pr) describes the relationship between momentum diffusivity and thermal diffusivity.

$$Nu_x = \frac{3}{4}\left(\frac{\Pr}{2.435 + 4.884 \cdot \Pr^{\frac{1}{2}} + 4.953 \cdot \Pr}\right)^{\frac{1}{4}} Ra_x^{\frac{1}{4}}. \tag{4}$$

The convective heat transfer coefficient (h_c) is calculated as

$$h_C = \frac{Nu_x \cdot \lambda}{x} \tag{5}$$

The fluid thermal conductivity (λ) and vertical position on the radiator surface are included when the heat transfer coefficient is determined. Air data for ν, α and λ at different water supply temperatures are presented in Table 2.

Newton's law of cooling [15] expresses the heat transfer rate due to convective heat transfer as

$$\dot{Q} = h_c \cdot A \cdot (T_w - T_{ref}) \tag{6}$$

where (A) is the area of the surface.

Table 2
Data for different supply temperatures.

Water supply temperature (°C)	Mean difference temperature (°C)	ν (m²/s)	α (m²/s)	λ (W/m·K)
75.0	47.5	$17.97 \cdot 10^{-6}$	$27.83 \cdot 10^{-6}$	$25.56 \cdot 10^{-3}$
55.0	37.5	$16.97 \cdot 10^{-6}$	$27.09 \cdot 10^{-6}$	$24.08 \cdot 10^{-3}$
45.0	32.5	$16.46 \cdot 10^{-6}$	$26.72 \cdot 10^{-6}$	$23.34 \cdot 10^{-3}$

Download English Version:

https://daneshyari.com/en/article/644782

Download Persian Version:

https://daneshyari.com/article/644782

Daneshyari.com