ELSEVIER

Applied Energy 65 (2000) 251–256

APPLIED ENERGY

www.elsevier.com/locate/apenergy

Design and simulation of a new energy-conscious system (CFD and solar simulation)

Mohamed B. Gadi*

School of the Built Environment, The University of Nottingham, University Park, Nottingham NG7 2RD, UK

Abstract

This paper presents the use of a validated CFD programme (FLUENT) and a solar simulator, for designing a solar water-heater. The water-heater is part of a new passive cooling and heating system introduced for buildings in North Africa. CFD transient simulations were carried out using a small time-step of 10 s and a set of fine body-fitted computational grids (1770–4740 nodes). FLUENT results were then verified against indoor testing employing a solar simulator. Good agreement was achieved. © 1999 Elsevier Science Ltd. All rights reserved.

Keywords: CFD; Fluent; Solar simulator; Water-heater

1. Introduction

Computational fluid dynamics (CFD), is a powerful simulation technique. It has been used to simulate wind effects on building envelopes, indoor air movement, temperature distribution within buildings and performance of heating and cooling systems. The CFD software (FLUENT) employed in the present work was actually used as a design tool. It was developed by Fluent Europe in the UK and Fluent Inc. in the USA. Holmes in 1982 [1], used the computer code PHOENICS to study the influence of indoor structural elements such as ceiling beams on wall jets within a room. Awbi and Setrak in 1986 [2] used the TEACH code to investigate the effect of ceiling obstructions on wall jets and the associated velocity profile. In June 1994, Awbi published the results of a CFD simulation of air movement, temperature distribution and their effects on human thermal comfort in an atrium building using the VORTEX programme [2,3]. Thermal comfort was presented in terms of PMV values

* Tel.: +44-115-951-3118; fax: +44-115-951-3159.
E-mail address: mohamed.gadi@nottingham.ac.uk

PII: S0306-2619(99)00121-X

as predicted by Fanger's model. The influence of computational parameters in CFD simulation of wind environment around buildings was investigated by Baskaran [4]. CFD has also been used to study the performance of a solar chimney [5]. Boundary conditions for the CFD simulation were obtained from indoor testing using a model of a building with a solar chimney and a solar simulator.

2. FLUENT simulation of the water-heater

The procedure, by which a detailed configuration of the solar water-heater was developed, combined the use of architectural thinking, CFD simulation and indoor solar verification. It was generally an experimental technique in which a particular design was simulated and its features were observed during simulation. Observations were then employed to apply further development on configuration. It was, however, a very lengthy process, and only the final case is presented and discussed here. The aim of simulations was to develop a solar water-heater which is easy to manufacture and produces good heat extraction and dissipation. For the container to be easily manufactured, meant it had to be of the appropriate size. Good heat extraction and dissipation meant using finns and a flexible shape that allowed for internal hot water currents to move easily within the container. The rectangular shape with right angles was thought to induce stratification. After a number of fluent simulations and careful observations and a set of "sketches" supported by hand-made models, the final shape of the container was transformed into a parallelogram (Fig. 1).

The new shape was then planned for simulation within FLUENT, version 4.11. Three different BFC grids were established, which had four different boundaries. In this case the container was simulated in its real size. The grids had the following number of nodes; ($59*30$, $73*52$, $79*60$). All simulations were performed in a transient mode with 10 s time steps. The maximum simulation time was 3600 s during which a fully converged solution was achieved after 360 time steps with 900 iterations for each time step (total 324,000 iterations).

Fig. 1 presents rasters of heat gained by the water after 1 h. In configuration (a) mixing is more visible although heat extraction is less, compared with other configurations. Configuration (b), seems to offer similar mixing but with higher heat extraction than case (a). In configuration (c), the heat flux boundary was simulated inside the container behind a glazing. In this case heat extraction has further increased stratification at the top of the container. Fig. 2 illustrates profiles of heat gained by the water after 1 h simulation, in which case (c) shows the highest level.

3. Validation of FLUENT results using a solar simulator

At this stage a full size model of the container was built from galvanised steel sheet and inserted into an insulated box. Making of the model, as noted by the technician who carried out the job, was quite easy. Based on the drawings and a small hand-made model provided by the author, the technician then cut the required flattened