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

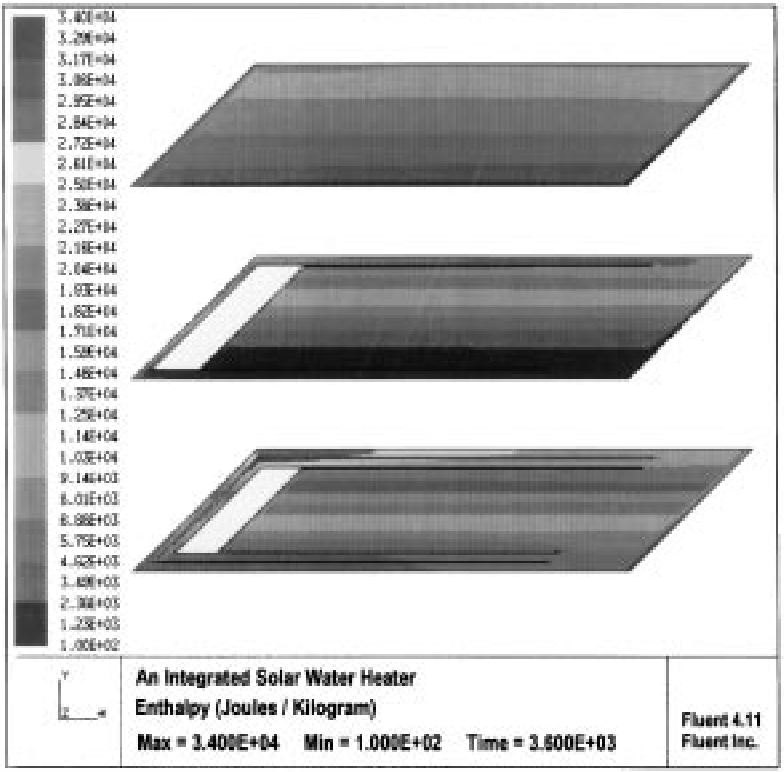

Fig. 1. Thermal performance of the solar water-heater designed as part of the Gadi system. Results of CFD simulation after 1 h of exposure to a constant heat flux (800 W/m^2).

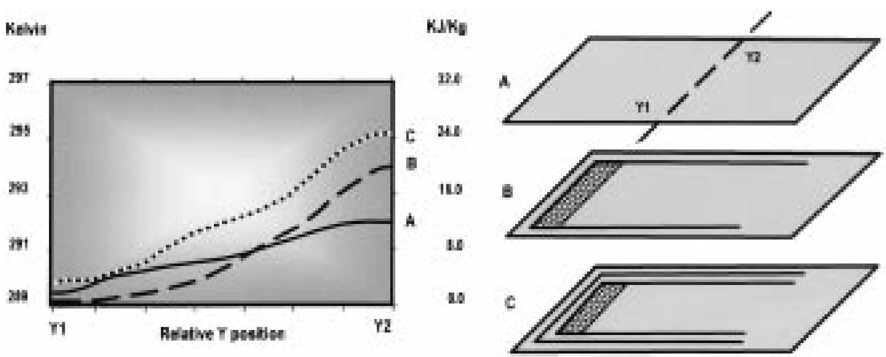

Fig. 2. Profiles of water temperature (K) and heat gain (kJ/kg) after 1 h, FLUENT simulation.

Fig. 3. Connecting the K-thermocouples to the water-heater.

Fig. 4. Solar simulation of the thermal storage component in the new Gadi system.

form of the container on a sheet of steel. The cut sheet was then folded and welded from the back and one side. The front cover was cut separately from a copper sheet for better heat conduction. The cover was later bolted to the container using steel screws and rubber seal for water tightness and to allow for possible expansion of the cover. Indoor testing was carried out using a solar simulator. Temperature measurements were recorded through a number of thermocouples (type K) at locations on the container surface and within the water (Fig. 3). To insure that no heat flux

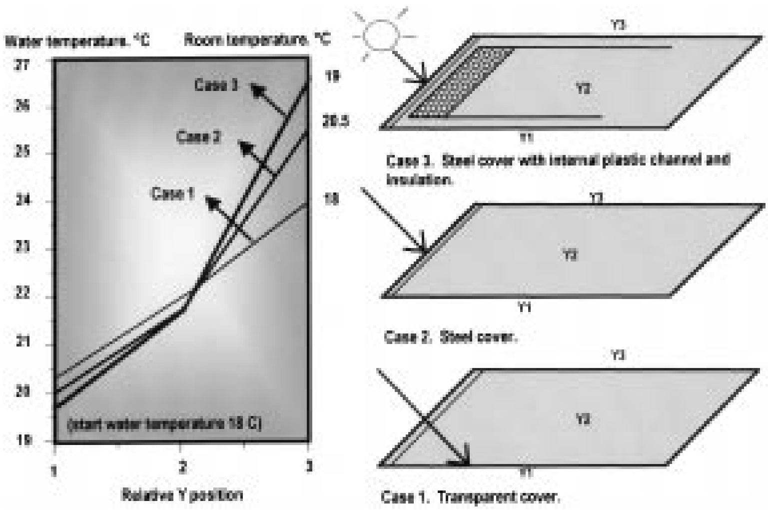

Fig. 5. Thermal performance of the solar water-heater designed as part of the Gadi system. Results after 1 h of exposure to a constant heat flux (800 W/m^2).

would reach the water except through the collector plate, all exposed metal parts of the container were covered with aluminium foil (Fig. 4). Precautions were taken at the beginning of measurements to insure good mixing and uniformity of temperature distribution in water. Because of water leakage problems encountered during the experiment, which required overnight treatment, the results were gathered on three different occasions and consequently within three different ambient temperatures. This was due to lack of control over the test room temperature which was linked to a central heating system. Indoor simulation was performed on three configurations of the water container. One configuration was for the container without any internal components. A second configuration was for the container with an internal unit made from plastic and an air gap as an insulation. A third configuration which was similar to the first configuration except for its glazed cover. Results shown in Fig. 5 illustrate good agreement with FLUENT predictions given in Fig. 2, with regard to direction of data points. As in FLUENT results, case three with internal partition and insulation gave a higher heat extraction and, therefore, temperature when compared with case 2. Case 1 was included in the experiment for comparison and to show the effect of the collector.

References

[1] Holmes MJ. The application of fluid mechanics simulation programme PHOENICS to a few typical HVAC problems. London (UK): Ove Arup and Partners, 1982.
[2] Awbi HB, Setrak AA. Numerical solution of ventilation air jet. 5th ed. Symposium on the use of Computers for Environmental Engineering Related to Buildings, UK: CIBSE, 1986.

[3] Awbi HB, Bazihan L. Predicting the thermal and airflow performance of large spaces. ROOM-VENT'94, vol. 1, Air distribution in rooms. 4th. Int. Conf., Krakove, Planed, 1994.
[4] Baskaran A, Stathopoulos T. Influence of computational parameters on the evaluation of wind effects on the building envelope. Building and Environment 1992;27(1).
[5] Barozzi GS, Imbabi MSE, Nobile E, Sousa ACM. Physical and numerical modelling of a solar chimney-based ventilation system for buildings. Building and Environment 1992; 27(4).