Available online at www.sciencedirect.com
SCIENCE DIRECT
Energy and Buildings 35 (2003) 893–902
ENERGY and BUILDINGS
www.elsevier.com/locate/enbuild

Application of CFD in retrofitting air-conditioning systems in industrial buildings

Shengwei Wang [a,*], Dongsheng Zhu [b]

[a] Department of Building Services Engineering, The Hong Kong Polytechnic University, Kowloon, Hong Kong
[b] The Key Laboratory of Enhanced Heat Transfer and Energy Conservation, South China University of Technology, Guangzhou, PR China

Received 18 February 2002; accepted 24 December 2002

Abstract

This paper presents a practical case study using the computational fluid dynamics (CFD) technique in retrofitting the large cabinet fans and airflow channels in malt processing air-conditioning systems in order to get uniform airflow field, to increase fan capacity and reduce energy consumption.

Site measurements were used to validate the models before they were used. CFD simulation results were used to evaluate and optimise the design and construction of splitter vanes in the airflow channels and scrolls of the fans. Simulation tests showed that the fan capacity (maximum total air flow rate delivered at full fan speed) could be increased by 8.3–20% using different shapes of scroll, and 30–34% by adding splitter vanes further in the airflow channels. After retrofitting the fans and airflow channels, site measurements showed that the fan capacity increased 21–24 and 28–29%, respectively in the two buildings validated. The approach of using the CFD technique in the retrofitting, the simulation results, and the site measurements before and after fan retrofitting are presented and compared in this paper.

Keywords: Computational fluid dynamics; Computer simulation; Fan; Air-conditioning; Retrofitting; Energy conservation; CFD

1. Introduction

The malt bed air-conditioning system is the main energy consumer in a malthouse. To efficiently remove the heat from malt beds, uniform distribution and sufficient cold air flowing through the malt beds are required. In large industrial buildings, the fans used in the air-conditioning systems are rather big, and their energy consumption is a significant part of the production cost. It is even more important to have adequate fan capacity (maximum total air flow rate delivered at full fan speed) and even airflow distribution through malt beds to ensure the quality of production. Effective cooling of malt beds and energy saving are among the main concerns in operating the air-conditioning systems in malthouses. In particular, due to the hot weather in summer in south China, there are often not enough airflow rates in the systems to cool the malt beds.

There are two usual approaches to improving the airflow distribution and saving energy. The first uses experimental tests [1] and the second uses numerical simulation [2]. The latter has many advantages in application, such as flexibility in shaping the flow channel, especially for large spaces and complex geometric corners of buildings. Furthermore, the cost of computation tests decreases continuously while labour and materials costs rise.

Computational fluid dynamics (CFD) [3] has successfully been applied in the aerospace [4], HVAC [5–8] and food processing industries [2], among others. The advantages of using numerical methods to analyse airflow fields have been summarised by Fletcher et al. [5]. In particular, the predictions obtained by numerical methods have enabled the concepts of ventilation efficiency to be applied at the design stage, while the value of the experimental method is restricted to evaluation and diagnostic studies on existing systems. Chung and Dunn-Rankin [8] presented a numerical simulation for predicting ventilation efficiency in a model room. Peng and Paassen [9] developed a state space model for predicting and controlling the temperature responses of indoor air zones. Aganda et al. [10] presented their study using CFD on the velocity and temperature distribution in a room heated by a warm air stream introduced at various levels. Sinha et al. [11] applied CFD to simulate the two-dimensional room airflow with and without buoyancy. Chou and Chang [12] presented a generalised methodology for determining the annual total heat gain through the

* Corresponding author. Tel.: +852-27665858; fax: +852-27746146.
E-mail address: beswwang@polyu.edu.hk (S. Wang).

0378-7788/03/$ – see front matter © 2003 Elsevier Science B.V. All rights reserved.
doi:10.1016/S0378-7788(03)00010-0

external walls and roofs of large air-conditioned buildings. The methodology can be applied to different classes of buildings, construction types and locations. Khouja et al. [13] applied a simulation model to assess the economics for precision manufacturing of air-conditioning equipment in order to evaluate the benefits of energy saving, dealing with possible correlations among the benefits of innovative cases.

This study concerns the use of computational fluid dynamics in retrofitting existing air-conditioning systems in food industrial processing buildings. Computational fluid dynamics was employed in this study to simulate the flow field inside the buildings when different retrofitting alternatives are implemented in the systems. A retrofitting design was chosen and adopted in the buildings to improve the air-conditioning systems. Using CFD simulation to predict the flow capacity and airflow patterns, the performance and benefits resulting from actual retrofitting, if different retrofitting alternatives are applied, were evaluated.

This paper presents an approach using CFD simulation in retrofitting, the simulation results and the actual performance of the air-conditioning systems after retrofitting, and a comparison between the simulated and measured system performances before and after retrofitting.

2. Air-conditioning system and site measurement

2.1. The air-conditioning systems and problems

This study was concerned with buildings for malt production. The malthouse involved was one of the largest in Asia when it was built. There were 12 malt processing systems (i.e. malt beds) in the malthouse. Six of the beds used in this study were identical. The systems had been used for over 10 years. Since the original design was based on the European climate, the cooling capacities of the systems were not sufficient for use in the tropical climate of Guangzhou (China). The main problems of the systems were the insufficient airflow and the uneven distribution of the airflow through the malt beds and the evaporators. This resulted in the uneven temperature of the malt on the malt beds. Consequently, the capacity of the evaporators was not efficiently used, and it was therefore difficult to control the quality of the products.

A cross-section of the buildings is illustrated in Fig. 1. It consists of a fan (#1), an evaporator room (#2), the evaporator (#3), a room below the evaporator (#4), a room below the malt bed (#5), the malt bed (#6), the malt bed room (#7) and three air shutters (#8, #9,#10). The fan used had a design power of 60 kW and a diameter of 2 m. The evaporator was 4.5 m × 3 m × 0.6 m (Length × Width × Height) and the malt bed has a diameter of 25 m.

As shown in the figure, the rooms used as the airflow channels contained many abrupt enlargements, sudden contractions and rectangular elbows, as the airflow circulated in the building without air ducts. In the production process, the fresh air and re-circulated air were mixed before entering the fan cabinet. The airflow was driven into a large space—the evaporator room, where the airflow channel was abruptly enlarged. The airflow then passed the evaporator, a contraction entrance in which the air was cooled before entering another large space (room #4). The airflow from the evaporator suddenly changed direction in this room and then entered the room (#5) below the malt bed after an abrupt enlargement. The airflow changed direction again in this room

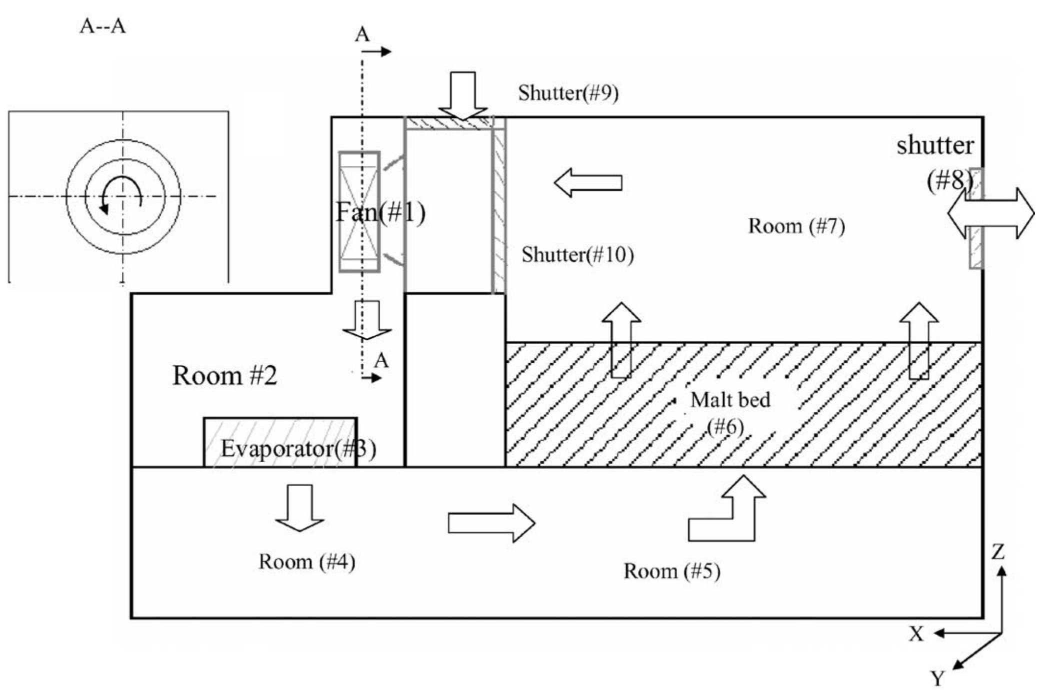

Fig. 1. A (vertical) cross-section sketch of the malt building, top right is a cross-section sketch of the fan.

before passing through the malt bed. The fresh air was taken from the shutters (#8, #9) when beneficial (i.e. when the outdoor air temperature was lower).

The numerical test results were conducted to provide guidance for improving the construction of air channels inside the buildings in order to increase the fan efficiency and capacity and to obtain a uniform air velocity and pressure drop on the malt bed. In particular, it is advantageous to use CFD to analyse the airflow pattern inside the fan chamber and airflow channels [14,15]. In this case, the simulation test initially provided the evaluation with fruitful information on the significance of retrofitting, and the owner was therefore persuaded to invest in the retrofitting project. In the actual retrofitting design process, the simulation allows a large number of tests to be conducted, which is costly and impractical in terms of time, and the best retrofitting design can be achieved within a very short time. Therefore, efforts were made to simulate the airflow pattern to obtain an optimal option for fan and airflow channel retrofitting in the buildings.

2.2. Cabinet centrifugal fan system and site measurements

A typical centrifugal fan consists of a rotary impeller with a fixed scroll [16]. Unhoused centrifugal fans do not have the outer casing or scrolls. The impeller is mounted in a cabinet. They are often called cabinet fans. The fans used in this malthouse were cabinet fans. The whole space (the four walls and the roof) of the fan chamber was, in fact, the scroll of the fan as shown in Fig. 1(A-A). The diameter of the impeller was 2 m, and the size of the chamber was $4.2\,\mathrm{m} \times 3\,\mathrm{m} \times 0.8\,\mathrm{m}$ (Length × Height × Width). However, the fan chamber was too long and wide, resulting in large vortices in the corners. In fact, it did not meet the basic design requirement that the flux channel of the scroll be extended gradually. Therefore, it was decided to design and construct a scroll around the impeller (Table 1).

To accurately evaluate the effects of fan retrofitting, it is essential to measure the operation parameters of the air-conditioning system and the airflow distribution in the buildings [5]. The measurement procedure was carefully designed to measure the fan speed and power at the same voltage, and the fan capacity (total air volume flow rate) at the same stage of the malt production cycles, in order to compare the actual fan performance before and after retrofitting on the same basis.

It is also important to measure the air velocity distribution, airflow rate and other operation parameters to validate the CFD simulation models and to assess the problems of airflow distribution. Two different types of meters were used to measure the air velocity and volume flow rate to ensure the accuracy of measurement by comparing the readings of the two meters. One was an “airdata multimeter” made by an American firm, and the other was an “electronic breeze instrument”, a kind of hot wire air velocity meter, made in Tianjin. Both had a minimum scale division of 0.01 m/s and a measurement error of less than 2%. The air velocity at the fan entrance (ϕ 1.4 m) was measured at 132 (11×12) points evenly distributed on the cross-section. A frequency inverter was installed to control the fan speed, so that the fan capacity at different speeds could be measured.

Table 1
Parameters of fan chamber before retrofitting

Size of fan chamber (m)		Parameters of fan	
Length	4.2	Diameter (m)/width of impeller (m)	2/0.45
Width	0.82	Nominal power of fan (kW)	60
High	3	Design efficiency of fan (%)	42

3. Application of computational fluid dynamics

3.1. Outline of CFD models

Application of CFD in simulating fluid flow and heat transfer has been steadily increasing over the last 30 years [3,17,18], particularly since the development of the software packages with user-friendly interfaces. A commercial CFD software, PHOENICS, was used in this project. The general-purpose software package, PHOENICS [19], solves the N–S equations to predict quantitatively how fluids (air, water, oil, etc.) flow in and around engines, process equipment, buildings, natural-environment features, etc.

Based on the assumption that the airflow is incompressible and steady-state in the air-conditioning system in the buildings, the governing equations for the general dependent variable, ϕ, can be expressed by the following general equation:

$$\operatorname{div}(\rho\vec{u}\phi) - \operatorname{div}(\Gamma_\phi \operatorname{grad}\phi) = S_\phi, \tag{1}$$

where ρ is the density, S_ϕ the source/sink rate per unit volume for the dependent variable ϕ, and Γ_ϕ is the effective exchange coefficient of ϕ.

The airflow domain is bound by the impeller passage, inlet and outlet sections, and outer walls of the fan chamber (before retrofitting), or the new scroll (after retrofitting). The geometry of the fan chamber, the proposed scroll and the wall of the air channels (i.e. the rooms), as well as the boundary conditions applied, are properly represented in the mathematical models using the graphic interface provided by PHOENICS. The boundary conditions are set as: (a) zero velocity ($u = v = w = 0$) at all wall surfaces; (b) volume velocity is $40\,\mathrm{m^3/s}$ at the fan chamber inlet; (c) pressure is 0 Pa at the fan chamber outlet.

The computational procedure adopted for such a three-dimensional turbulent airflow is based on solving the governing equations for the dependent variable, which consists of the three velocity components, and the pressure by means of the finite volume technique.

The gradient of flow variables in a control volume is set to zero [18]. The velocity component is corrected at the inlet and outlet planes, so that the flow rate has a constant value. The wall function method is employed. The sliding mesh method is used in modelling the fan, as it is the most accurate method in simulating multiple moving reference frames although it is computationally demanding. The impeller inlet and outlet are assumed to be periodic surfaces, and periodic boundary conditions are given on these periodic surfaces [20,21]. As the geometry of such a flow configuration varies in a repeating manner along the flow direction, leading to a periodic fully-developed flow regime in which the flow pattern repeats in successive cycles, the use of periodic boundary conditions results in less expensive computation.

In order to solve the set of the above-mentioned differential equations, the finite-domain technique was used, incorporating the features of the methods proposed by Patankar and Spalding [17]. The space dimensions were discretized into finite intervals and the variables were computed at a finite number of grid points. This paper utilises the so-called LVEL turbulence model [20,21]. The LVEL model is regarded as a suitable method providing a practical solution to airflow problems. These variables are interconnected through algebraic equations derived by integrating the differential equations over the control volumes defined by the above grid. This leads to the algebraic equations of the form shown in Eq. (2).

$$\sum_{n}(A_n^{\phi} + C)\phi_{\mathrm{p}} = \sum_{n} A_n^{\phi}\phi_n + CV \tag{2}$$

where the ‘Σ’ means the summation over all the cells adjacent to a certain defined point. The coefficients, A_n^{ϕ}, accounting for connective and diffusive fluxes across the elemental grid cells, are formulated using hybrid differencing. The present model was implemented in PHOENICS (version 3.2).

The solution method developed in this study made use of the LVEL model provided by PHOENICS. All models were implemented through the computational framework of PHOENICS. The calculations were conducted on a PC computer with a main memory of 256 MB.

3.2. Outline of CFD simulation test procedure

In the first stage of the simulation study, computation trials using various intervals and numbers of grid points were tested. The simulation results, i.e. the total flow rates and the patterns of the velocity vector inside the fan chamber, rooms (#1, #2, #4, #5) and equipments (#3, #6) were compared with the total flow rate and flow distribution measured on site in selected sections and critical locations.

Non-uniform grids were used in both the fan chamber and the rooms. All the cells used in the rooms were rectangular cells, while some grids used to define the cells in the fan chamber were arc-shaped. A number of tests were conducted to find out the suitable numbers of grid points providing grid-independent solutions. Eventually, the number of grid points used in simulating the fan chamber was around $30 \times 90 \times 70$ in the x, y and z directions, respectively. The number of grid points used in simulating both rooms without splitter vanes was around $72 \times 86 \times 92$ in the x, y and z directions, respectively. When simulating the rooms with splitter vanes, the number of grid points was slightly increased. The actual grid numbers in each test varied slightly concerning the solution and the speed of the simulation computation.

In the second stage of the simulation study, simulation results when the fan scrolls of different sharps and splitter vanes at different locations were used, i.e. the total flow rates and patterns of the velocity vector, were studied and compared. Over 300 simulation tests were conducted during this stage in order to find proper designs for the fan scroll and splitter vanes.

4. Results and analysis

A few simulation results before and after implementing the selected design of the fan scrolls and splitter vanes in the buildings are presented below for illustration and comparison. The simulation results are also compared with the data measured in situ before and after retrofitting the systems.

4.1. Results of simulation and problems before retrofitting

Fig. 2a shows the simulated velocity vector, which describes the velocity direction of every calculating point in the flow field, on the z–x plane (at $y = 3.2$ m) in the cabinet fan chamber and the evaporator room. There was a swirling motion at the top (location B) of the fan chamber. The simulated air flow field, which is used to display velocity isolines reflecting the velocity gradient of air flow on the same z–x plane is shown in Fig. 2b. There were some big eddies in the top corners of the fan chamber, causing a sharp reduction of air velocity at the outlet of the impeller. These eddies seriously affected the velocity flow field in the fan chamber, resulting in low fan efficiency. There were a lot of multi-direction flows, including turbulent flow, and eddies between the fan wheel and the wall. This indicated that the upper space above the fan, and the gap between the fan wheel and the wall, needed to be reduced.

Two very large swirling motions can be observed in the evaporator room: a swirling motion in the right corner (location A), and another swirling motion in the left corner (location C). Fig. 3 shows the simulated velocity vector in the evaporator room on the other planes. It again indicated the same phenomenon: a swirling motion created in two corners (location A and C) of the evaporator room. In Fig. 3b, two eddies (location D and E) can be observed around the walls. There were also eddies observed in the room below the evaporator.

The results of CFD simulation show that there were a lot of swirling motions created in the two top corners of the

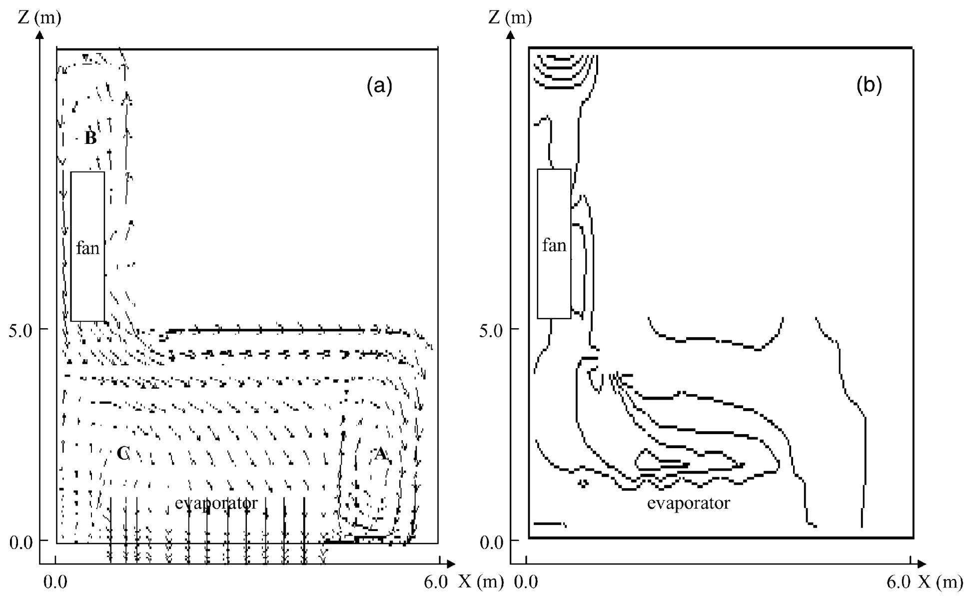

Fig. 2. Simulated velocity of airflow (a) and airflow field (b) at the z–x plane ($y = 3.2$ m) in the evaporator chamber before retrofitting.

fan chamber due to the abrupt enlargements at the impeller outlet (upper half) and the contraction at the chamber outlet. In this case, flow separation occurred, and eddies and large-scale turbulences, both after the enlargement and before/after the contraction, were created. Both caused losses in total pressure. To reduce such energy loss, the cabinet fan needed a gradual expansion channel, i.e. scroll. However, the geometric constraints of the fan chamber made it impossible to use a traditional whorl scroll, since the whole space below the impeller was the outlet of airflow directly connected to the evaporator chamber. A scroll of a diffuser angle of enlargement, as shown in Fig. 4, was chosen and designed according to the site geometric constraints and the best airflow distribution and total airflow rate given by the CFD simulation tests. A number of plates were added in the rooms above and below the evaporator as splitter vanes to eliminate the swirling motions and eddies.

4.2. Simulation tests for retrofitting and improvement

Various geometries of scroll were implemented in simulation tests to evaluate their effects on the fan performance (i.e. fan capacity). Fig. 5 presents the simulated fan flow capacity using five (more were tested in the case study) different scroll designs at three different fan speeds. In the figure, the CFD simulation results were compared with the measured performance data in one of buildings. The fan scroll design was modified and adjusted to provide good performance and match the constraints of the fan chambers. The scroll, #5, presented the best performance among the scrolls tested. The improvement of flow capacity when using this scroll design was between 21 and 23%. However, due to the difficulties of site installation, it was not practical to install scroll #5 in the buildings. Scroll #4 was selected since it provided significant improvement (between 19% and 20%)

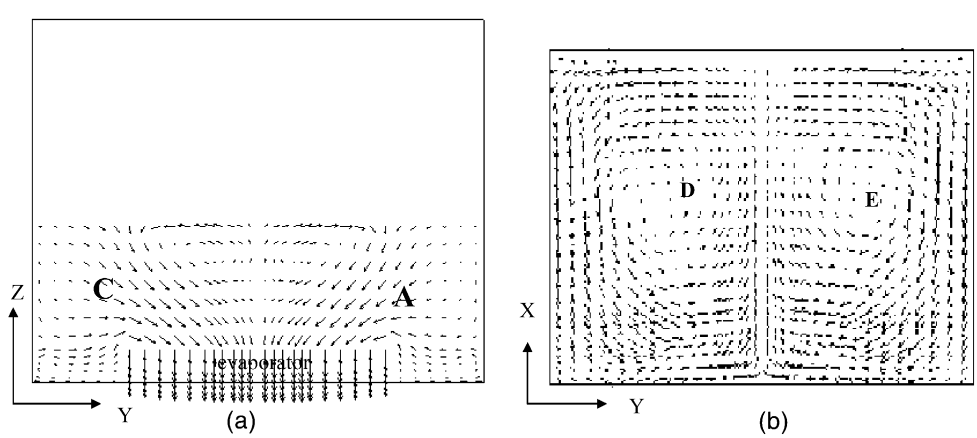

Fig. 3. The simulated velocity vector in the evaporator chamber. (a) In the z–x plane-elevation; (b) in the x–y plane-plan form.

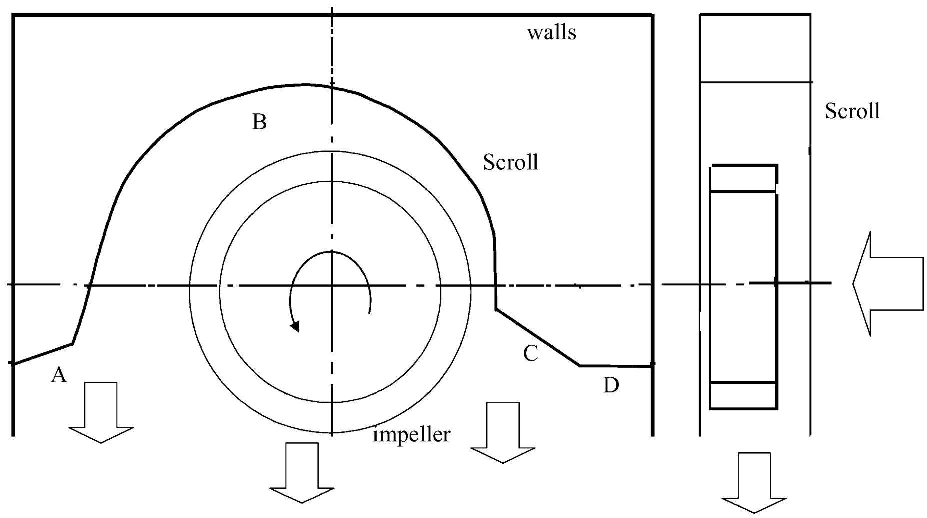

Fig. 4. A cross-section sketch of the fan chamber with the scroll designed.

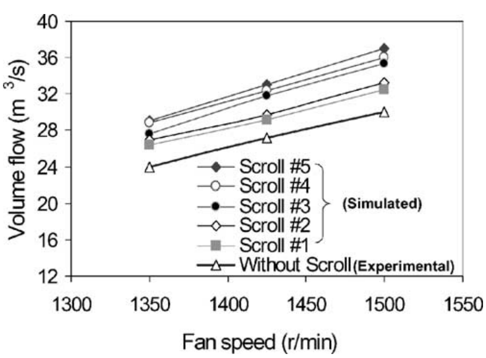

Fig. 5. The simulated system volume flow rate vs. fan speed when adding different scrolls.

on the fan flow capacity and was conveniently and economically installed in the buildings.

Fig. 6 presents the simulated velocity vector and airflow field after the fan was added with the selected scroll. It can be observed that the airflow distribution was improved and the swirling motions inside the fan chamber and the room below the fan were eliminated. The increase in the line density in the airflow field diagram also indicates that the airflow velocity increased, as an increase in the line density means an increase in the velocity gradient.

Fig. 7 presents an example showing the effect of splitter vanes used in eliminating the swirling motions in the room below the evaporator. The simulated velocity vector in a z–x plane (elevation) is presented. Fig. 7a presents the velocity vector without plates P5 and P6. Fig. 7b presents the velocity vector when plates P5 and P6 were installed. There were two eddies (F and G) at the airflow outlet of the evaporator before the plates were used. The airflow passed the evaporator, a sudden contraction entrance, and then suddenly entered a large space, where the airflow turned at a rectangular angle. In Fig. 7b, it can be observed that the use of the two plates eliminated the eddies.

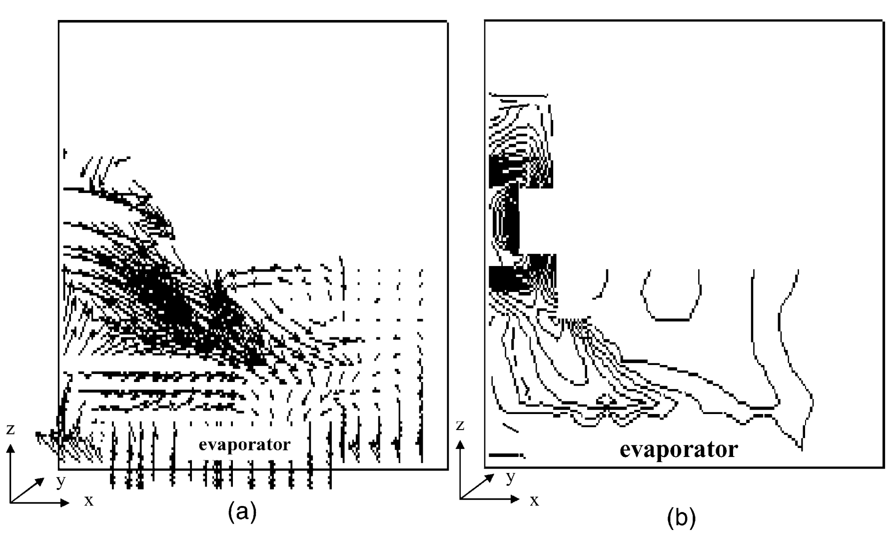

Fig. 6. The velocity vector of the airflow (a) and the airflow field (b) in the evaporator room (at $z = 0.8$ m) after fan retrofitting.

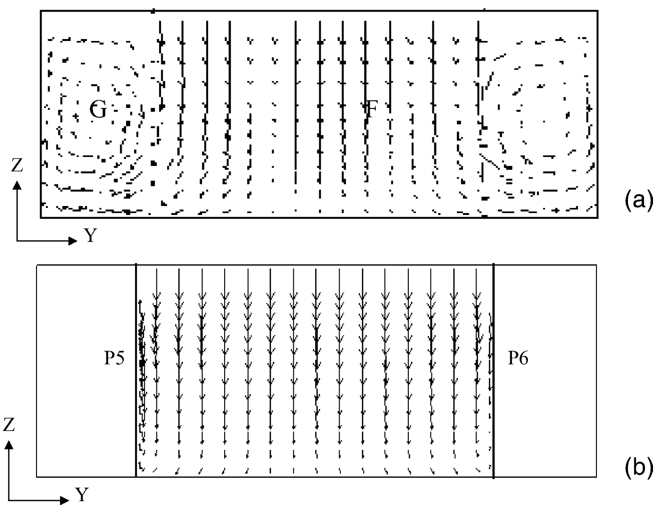

Fig. 7. Simulated velocity in the room below the vaporator in the z–y plane (elevation) before (a) and after (b) installing plates P5 and P6.

Various alternative options (i.e. geometries and locations of splitter vanes) were evaluated by CFD simulation concerning the airflow pattern and total flow capacity. Another major concern was to use simple splitter vanes to ensure the cost competitiveness of the retrofitting project. Eventually, eight simple plates were selected and installed in the two rooms as shown in Fig. 8. Plates P5 and P6 kept the air channel area unchanged after the airflow exited the evaporator (entering room #4). After the addition of plates P7 and P8, the reverse air stream was reduced. These four plates installed in the room below the evaporator eliminated the large eddies in the room and allowed the airflow to reach the malt bed with much more uniform distribution.

Fig. 9 shows the simulated volume flow capacity at different fan speeds after different splitter vanes were installed. When plates P1–P4 (the four splitter vanes around the evaporator in the evaporator room), the volume flow capacity increased by 9.0–9.7%. When plates P5–P8 were added in the room below the evaporator, the volume flow capacity achieved a further increment of 3.1–5.3%.

Simulated results also show that the large eddies disturbed the airflow, resulting in extra consumption of the fan power and low volume flow capacity, and these large eddies were eliminated or significantly reduced after the splitter vanes were introduced. Simulation results show that the volume flow increased 11–14% when eight splitter vanes were added to the rooms.

4.3. Site assessment of system performance

The above designs of fan scroll and splitter vanes were selected according to the simulated performance of the fan flow capacity and flow patterns while considering the site, the technical constraints, and the cost of the retrofitting work [22,23]. The simulation tests showed that the proposed retrofitting could provide an improvement in the flow capacity of up to 34%, that the economic benefits of retrofitting the air-conditioning systems were very significant, and that the payback period was less than 1 year. The management was, therefore, convinced to retrofit the systems in two buildings (#1 and #2) initially as trial cases. The performance of these two systems was carefully monitored before and after retrofitting, and the actual improvement on the flow capacity was up to 24 and 29%, respectively. Eventually, the same retrofitting on the four systems of same type was conducted. The performance of the first two systems moni-

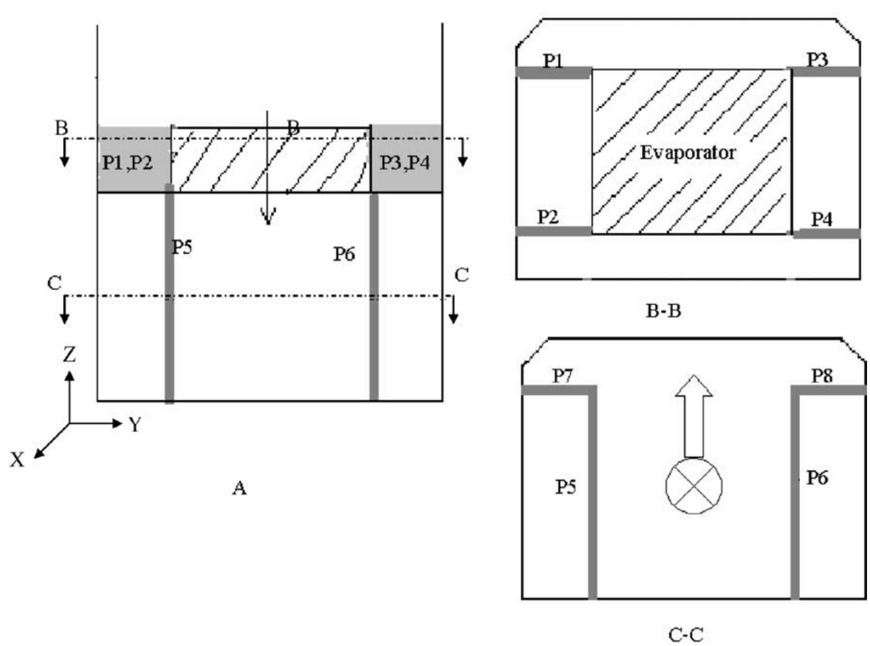

Fig. 8. Illustration of designed splitter vanes installed in the evaporator room and the room below the evaporator.

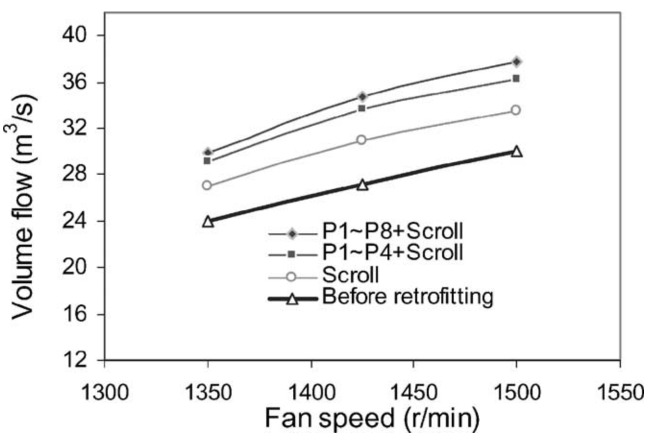

Fig. 9. Simulated total volume flow rate vs. fan speed before and after adding the fan scroll and splitter vanes.

tored before and after retrofitting is presented and compared below.

Figs. 10 and 11 present the measured flow capacities of the two systems before and after retrofitting and the comparison with the performance data given by the CFD simulation tests. The improvement (Fig. 10) of the fan retrofitting in building #1 was between 12 and 14%, which was obviously lower than the simulated results (between 19 and 20%). After the splitter vanes were installed in the building, the improvement was increased to 21–24% (simulated: 30–34%). The improvement when using the splitter vanes was 9–10% (simulated: 11–14%). It can be observed that the difference between the measured and simulated improvement of using the splitter vanes was obviously less than that using the scroll on the fan.

The improvement (Fig. 11) of the fan retrofitting in building #2 was between 14 and 16%, which was also lower than the simulated results. After the splitter vanes were installed, the improvement was increased to 28–29%, which was very close to the simulated data (30–34%). The improvement, resulting from retrofitting both the fan and the air channels in the second building, was more significant and much closer to the simulated results. The main reason observed was that installation in the second building was of a better quality and closer to the design intents, thus benefiting from the experiences learned during the work on the first building. The results from both buildings confirm that CFD simulation is a useful and efficient technical tool for selecting the retrofitting design of large airflow channels, and for making retrofitting decisions.

Site monitoring also showed that, when the scrolls were added, the fan power consumptions were increased slightly (normally by not more than 3%) at the same fan speed, while the flow capacity increased about 25% and much more uniform airflow distribution through both evaporators and malt beds was achieved. That effectively solved the problems of insufficient cooling capacity and uneven cooling of malt on the malt bed. As the heat exchange efficiency of the evaporator has been improved as a result of even air distribution, significantly higher airflow rate is, in fact, not needed in the air-conditioning systems. The fan can eventually operate at a lower speed. From site monitoring, a 20–28% saving in fan power consumption was normally achieved by operating the fans at lower speed.

In practice, the whole process of simulation and installation, including the management decision, of the fan scrolls and splitter vans in the first two buildings took around 1 year. That stage was completed in November 2000. The site test and validation took most of that time. The retrofitting of the other four buildings took about 4 months, and was completed in March 2001.

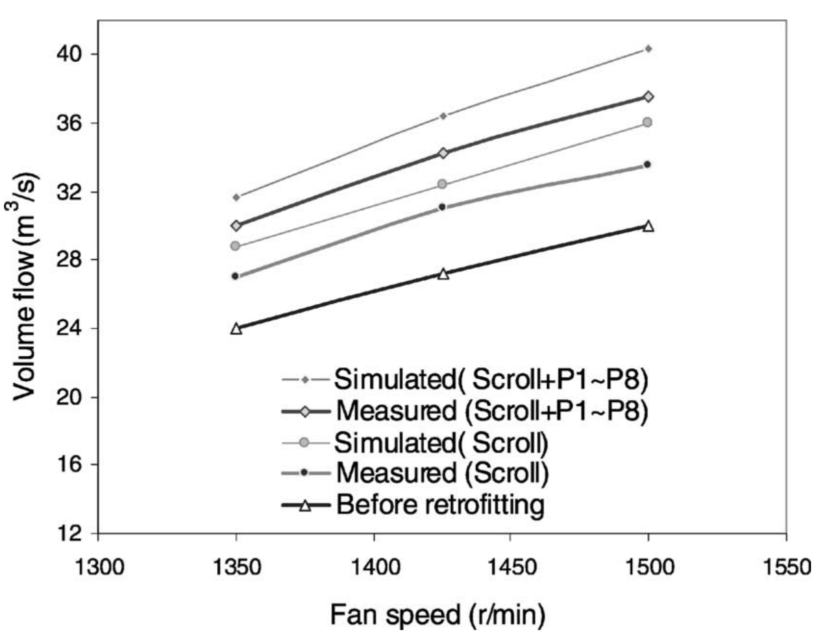

Fig. 10. Comparison between simulated and measured total air volume flow rate before and after installing the fan scroll and splitter vanes in building #1.

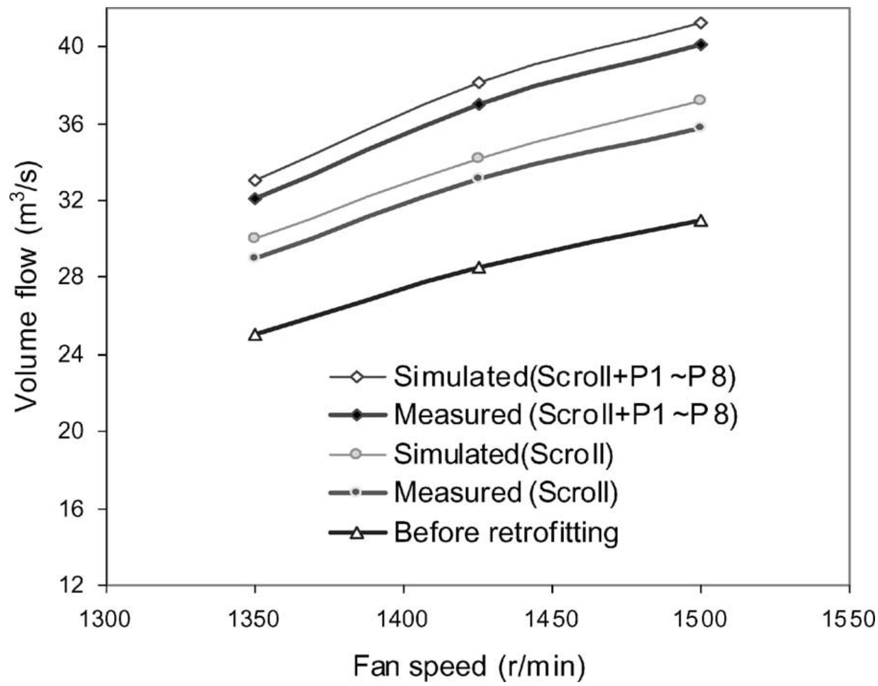

Fig. 11. Comparison between simulated and measured total air volume flow rate before and after installing the fan scroll and splitter vanes in building #2.

5. Conclusions

The retrofitting process demonstrates that CFD simulation can be used as an effective and economical tool to produce valuable guidance for practical improvements to the processing air-conditioning systems in industrial buildings. The CFD technique can reproduce the swirling motions and the positions of eddies, quantitatively confirming their dimensions. In retrofitting applications, CFD models can be checked using the measurements (i.e. flow distribution and total flow rates) of existing systems in practical application before the models are used to predict the performance after retrofitting. The accuracy and reliability of simulation results appears to be adequate for the practical applications. Certainly, serious validation of CFD models should be based on more fundamental measurements besides these overall parameters. This might not be practical, as cost and time are of major concern, and might not be necessary in practical applications like the study case, as requirements regarding computational accuracy are not very high.

The simulation and site studies show that significant improvements (up to 10–16%) can be made in the efficiency of cabinet fans in industrial air-conditioning systems by adding properly designed scrolls. The airflow pattern in large airflow channels can be improved by the use of certain simple properly sized and located splitter vanes that can improve flow capacity (up to 11–14%). Experience shows that the predictions on the effects of adding vanes to the airflow channels, simply by using the knowledge of fluid dynamics, often deviate dramatically from the data given by CFD simulation and on-site measurements. This confirms again the need for and value of numerical simulation tools in such retrofitting applications.

Another simple but important experience derived from the project is that the simulation outputs can give fruitful and convincing numerical details of the effects and benefits of retrofitting. This “numerical evidence” given by the CFD simulation tests can provide great support in convincing the managers to fund the retrofitting project.

Acknowledgements

This research work is financially supported by research grants from The Hong Kong Polytechnic University, the State Key Fundamental Research Program (no. G2000026305), and the National Nature Scientific Foundation of the PRC. The authors would also like to express their gratitude to the engineers of the Guangzhou Malthouse Ltd. for their technical support on the project.

References

[1] P.S. Mirade, J.D. Daudin, A new experimental method for measuring and visualising air flow in large food plants, J. Food Eng. 36 (1) (1998) 31–49.
[2] P.S. Mirade, L. Picgirard, Assessment of airflow patterns inside six industrial beef carcass chillers, Int. J. Food Sci. Technol. 36 (5) (2001) 463–475.

[3] S.V. Patankar, Numerical heat transfer and fluid flow, Hemisphere, McGraw-Hill, New York, 1980.
[4] G.W. Yang, L.X. Zhuang, Numerical simulation of rotor flow in hover. Simplified numerical simulation of transients on gas, Journal of Aircraft 37 (2) (2000) 221–226.
[5] C.A.J. Fletcher, I.F. Mayer, A. Eghlimi, K.H.A. Wee, CFD as a building services engineering tool, International Journal on Architectural Science V2 (3) (2001) 67–82.
[6] W.K. Chow, Application of computational fluid dynamics in building services engineering, Buildings and Environment V31 (5) (1996) 425–436.
[7] S.W. Wang, Y.M. Chen, A novel and simple building load calculation model for building and system dynamic simulation, Applied Thermal Engineering 21 (6) (2001) 683–702.
[8] I.P. Chung, D. Dunn-Rankin, Using numerical simulation to predict ventilation efficiency in a model room, Energy and Buildings 28 (1) (1998) 43–50.
[9] X. Peng, A.H.C. Paassen, State space model for predicting and controlling the temperature responses of indoor air zones, Energy and Buildings 28 (2) (1998) 197–203.
[10] A.A. Aganda, J.E.R. Coney, C.G.W. Sheppard, Airflow malt distribution and the performance of a packaged air-conditioning unit evaporator, Applied Thermal Engineering 20 (6) (2000) 515–528.
[11] S.L. Sinha, R.C. Aroraand, S. Roy, Numerical simulation of two-dimensional room air flow with and without buoyancy, Energy and Buildings 32 (1) (2000) 121–129.
[12] S.K. Chou, W.L. Chang, Generalised methodology for determining the total heat gain through building envelopes, International Journal of Energy Research 20 (10) (1996) 887–901.
[13] M. Khouja, J.W. Shelnutt, M. Wilmot, Economics of air-conditioning for precision manufacturing: a simulation model, Integrated Manufacturing Systems 9 (3) (1998) 182–192.
[14] M.B. Botros, D.F. Hanna, J.E. Boulos, M.C. Lai, New apparatus and blower centrifugal fan design features to improve its performance, ASME Fluids Engineering Division (Publication), FED 250, ASME, 14–19 November 1999, pp. 89–99.
[15] S.K. Wang, Handbook of Air-Conditioning and Refrigeration, McGraw-Hill, 1994.
[16] W. Lu, A.T. Howarth, A.P. Jeay, Prediction of airflow and temperature field in a room with convective heat source, Building and Environment 32 (6) (1997) 541–550.
[17] S.V. Patankar, D.B. Spalding, A calculation procedure for heat mass and momentum transfer in parabolic flows, International Journal of Heat Mass Transaction 21 (12) (1972) 1565–1579.
[18] C.K.G. Lam, K. Bremhorst, A modified form of the k-e model for predicting wall turbulence, ASME Journal of Fluids Engineering 103 (4) (1981) 456–477.
[19] D.B. Spalding, Phoenics Code Manual, London, UK, 1999.
[20] M.A. Leschziner, W. Rodi, Computation of strongly swirling an symmetric free jets, AIAA Journal 22 (12) (1983) 1742–1947.
[21] D. Agonafer, G.L. Liao, B. Spalding, The LVEL turbulence model for conjugate heat transfer at low Reynolds numbers, ASME 1993, PHOENICS Applications Album.
[22] W.C. Osborne, FANS, Pergamon Press, second ed., 1977.
[23] H. Manz, H. Huber, Experimental and numerical study of a duct/heat exchanger unit for building ventilation, Energy and Buildings 32 (2) (2000) 189–196.