OPTIMUM VENTILATION AND AIR FLOW CONTROL IN BUILDINGS

17th AIVC Conference, Gothenburg, Sweden, 17-20 September, 1996

(Title)

REDUCTION OF FLOW LOSS DUE TO HEAT RECOVERY IN PSV SYSTEMS BY OPTIMUM ARRANGEMENT OF HEAT-PIPE ASSEMBLIES

(Authors) S B Riffat and L Shao

(Affiliation)
Institute of Building Technology
University of Nottingham
Nottingham
NG7 2RD
United Kingdom

REDUCTION OF FLOW LOSS DUE TO HEAT RECOVERY IN PSV SYSTEMS BY OPTIMUM ARRANGEMENT OF HEAT-PIPE ASSEMBLIES

SYNOPSIS

Natural ventilation is being applied to an increasing number of new buildings to minimise reliance on mechanical ventilation and so reduce emission of greenhouse gases. However, passive stack ventilation (PSV) systems are currently designed without incorporating heat recovery leading to significant wastage of energy. Heat recovery systems have not been used in naturally-ventilated buildings because the pressure loss caused by a conventional heat exchanger is large compared to the stack pressure and could cause the ventilation system to fail. In addition, the stack pressure decreases owing to reduction of the temperature difference associated with the heat exchange, although this problem can be lessened by appropriate siting of the heat exchanger to maximise the effective stack height. In this study, natural convective flow through PSV stacks were computed using CFD to determine the effect of the layout of heat-pipe assemblies as well as the effects of spacing and length of fins on reduction of flow rate through the stack. Among the layout patterns examined, the arrangement where the assemblies are placed in a pattern of an arrow facing the flow direction produced the least insertion flow loss. The flow loss due to the insertion of the heat pipe assemblies (IFL) was found to increase sharply with the number of fins and reached over 30% when only 4 fins were used. IFL also increased with fin length but the rate of increase reduced for larger fin lengths. Therefore, for a given total surface area of fins, using fins with a larger length causes less flow loss than fins with a smaller spacing.

LIST OF SYMBOLS

g	Gravitational acceleration; 9.8 m/s^2
Gr	Grashof number; dimensionless
IFL	Insertion flow loss; %
l	Stack height; m
Q_{HP}	Volumetric flow rates in stack with heat-pipe assemblies; m^3/s
Q_{noHP}	Volumetric flow rates in stack without heat-pipe assemblies; m^3/s
T_w	Temperature of the stack walls; K
T_∞	Air temperature at the stack inlet; K
V_V	Vertical component of air velocity; m/s

1. INTRODUCTION

A wide range of buildings employ natural ventilation to minimise reliance on mechanical ventilation and so reduce emission of greenhouse gases. The passive stack ventilation (PSV) system which relies on the stack pressure created by the temperature difference between the indoor and outdoor air has been applied to various types of modern buildings, including offices, schools and houses [1, 2]. Natural stack ventilation consumes no power and so produces no harmful emissions, has no running cost, no noise of operation, requires little maintenance and because it involves no moving parts, operation is reliable. However, PSV systems are currently designed and constructed without incorporating heat recovery leading to wasteful heat loss. It has been estimated that this heat loss amounts to 3 - 15 GJ per annum for a small family residence and much more for larger buildings, e.g., offices [2]. There is obviously a need for appropriate and efficient heat recovery in PSV systems to minimise energy wastage. This paper presents results of a computer simulation which forms part of a feasibility study of a novel heat recovery system for use in PSV systems. The flow loss in the passive stack incorporating heat-pipe heat recovery was evaluated using computational fluid dynamics (CFD) to provide an insight into the mechanism and characteristics of the loss and an indication of its magnitude. Effects of spatial arrangement of heat-pipe assemblies and the spacing/length of fins were studied.

2. HEAT-PIPE HEAT RECOVERY SYSTEM

Heat recovery systems have not been used in naturally-ventilated buildings because the pressure loss caused by a conventional heat exchanger is large compared with the stack pressure and could cause the ventilation system to fail. In addition, the stack pressure decreases owing to reduction of the temperature difference associated with the heat exchange, although this problem can be lessened by appropriate siting of the heat exchanger to maximise the effective stack height. Research work on natural stack ventilation has been carried out by Schultz and Saxhof [3] using a counterflow heat exchanger consisting of channels separated by smooth corrugated metal sheets. This system had a high pressure drop, low capacity and large weight making it impractical to install in real buildings.

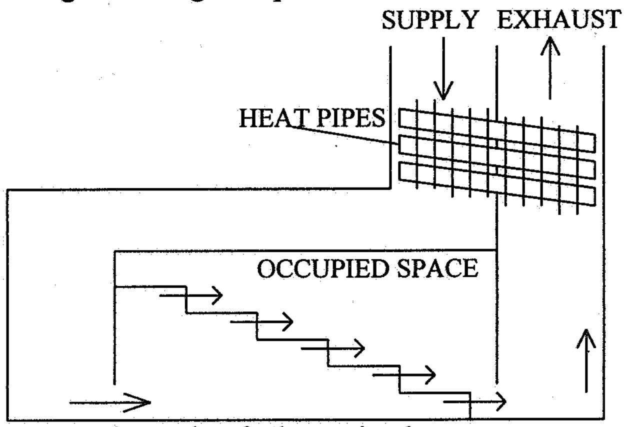

Figure 1. Schematic of a heat-pipe heat recovery system.

Heat pipes offer an alternative approach for heat recovery in naturally-ventilated buildings and have several advantages. The heat pipe consists of a sealed pipe lined with wick and partially filled with a working liquid. Installation is simple and involves locating one end of the tube in the warm exhaust ventilation stack and the other in the cool supply ventilation stack (see Figure 1). The liquid absorbs heat to evaporate at the warm end and condenses to release heat at the cool end, thereby performing the task of heat exchange. The condensed liquid is returned to the warm end by capillary forces in the wick fibres or by gravity. The heat pipe promises much higher capacity because it has much higher thermal conductance than the metal channels of conventional heat exchangers. In addition, it does not require complicated channels for supply and exhaust air. Individual heat pipes can be independently located in air ducts/cavities, making it easier to achieve a low pressure drop than using conventional heat exchangers which require the air path to be divided into narrow, flow resistant channels. Preliminary tests were carried out using a heat pipe system manufactured by Isoterix Ltd., UK. The results showed that for a heat recovery efficiency of 50%, stack flow rate of $0.056 m^3/s$ and stack flow speed of 0.5m/s, the pressure loss across an existing type of Isoterix heat pipe assembly is about 1 Pascal, which would not cause significant reduction of stack flow (The driving force of stack flow depends on temperature difference and stack height but is typically 5-10 Pascal). This pressure loss could be further reduced by adopting a more streamline shape for the heat pipes. The features described above make the heat pipe suitable for heat recovery in natural-ventilation systems.

3. DESCRIPTION OF THE COMPUTATION

The CFD code FLUENT was used to simulate pressure/flow loss in ventilation stacks by solving the Navier-Stokes equations, the mass conservation equation and the energy conservation equation. The simulated natural ventilation stack and the computational domain of this study is shown in Fig. 2. This study focuses on the buoyancy flow in the exhaust stack and the flow in the room connecting to the stack is not considered and therefore not included in the computational domain. Although the flow in the room may affect the flow in the ventilation stack by influencing the inlet boundary condition of the stack, this effect is dependent on individual room configuration and ventilation conditions, which vary considerably. Obviously, it is impossible to study all room configurations and difficult to select a small number of them as being representative of the vast range of room shapes. Therefore it was decided that only the stack flow should be examined. This is acceptable, as the buoyancy flow in the exhaust stack is the key to the functioning of a passive stack system. An understanding of the mechanisms of thermal and aerodynamic processes in the stack forms the basis for understanding heat recovery in natural ventilation systems. The above simplification of the computation domain is necessary because it reduces the demand on computing resources to a more acceptable level.

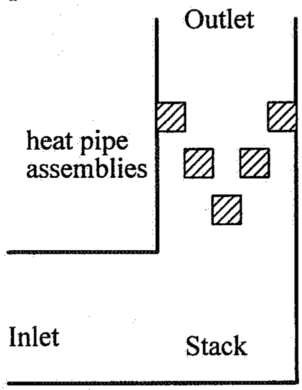

Figure 2. Schematic of the stack with heat recovery used in the simulation.

Computation of natural convective flow driven by small temperature differences such as those experienced in natural ventilation systems requires much more CPU time compared with computation of force flow in similar geometric set-ups [4]. In order to reduce the required CPU time to a manageable level, only two-dimensional computation was carried out. The inlet and outlet of the stack open to the outdoor atmosphere. The overall height and width of the stack (including the horizontal duct) are 3 m and 1 m, respectively. The width of the vertical stack and the height of the horizontal duct are both 0.4 m.

A major difference between the heat-pipe heat exchanger and more conventional heat exchangers is that for the former, heat is carried from the source to the sink by the refrigerants in the heat pipes instead of the wall separating the source and sink flows. To increase heat transfer capacity, the latter would resort to dividing the flow into smaller flow channels to increase the contact area while the former would simply require installation of multiple heat pipes spanning the source and sink flows. A typical heat-pipe has the shape of a cylindrical rod and several heat pipes are often bundled together and fitted with fins to form a heat-pipe assembly. As shown in Figure 2, heat-pipe assemblies are modelled in the simulation and placed perpendicular to the plane of the two-dimensional computational domain. The cross section of the assembly measures 0.08 m(w) × 0.09 m(h). It has been shown that the temperature of heat-pipe assemblies has little effect on the flow or pressure loss across the assemblies and therefore it was left at the default temperature (273K) set by the CFD code. As the fins are normally perpendicular to the heat pipes, they can not be simulated in a two-dimensional domain and the heat-pipe assembly would appear as a solid block. This limitation can be overcome by three-dimensional simulation, which will be carried out later. The limitation is not critical for this preliminary study, because the focus of the investigation is the loss of flow which would in any case normally flow around the assembly as if it were solid.

The computational domain is mapped by a Cartesian grid of 40×80 and the grid density in the axial direction of the stack is less than that for the spanwise direction to reduce the number of grid nodes required [5]. The walls of the stack have a temperature of 20°C and the inlet air temperature is 5°C. These were used to calculated the Grashof Number (Gr) to determine whether the flow is turbulent.

$$Gr = \frac{g(T_w - T_\infty)l^3}{T_w \nu^2}$$

where $g = 9.8\ m/s^2$, $Tw = 293$ K, $T\infty = 278$ K, $l = 3$ m and $\nu = 1.33 \times 10^{-5}\ m^2/s$. Based on the above values of the parameters involved, the resultant Gr is 7.75×10^{10}, which is greater than the transition value which is of the order of 10^8. Therefore the flow is turbulent and the k-ε turbulence model was used in the computation. The governing equations for the stack flow were discretised into finite volume equations, based on a Cartesian grid and the Power-law interpolation scheme. The equations resulting from the discretisation process were solved using the SIMPLE algorithm. Convergence acceleration methods including increasing the values of under-relaxation factors, multiple sweeps for the enthalpy equation and multigrid technique were used.

4. RESULTS AND DISCUSSION

Twelve cases of natural convective flow through the stack were computed, including cases without heat pipes and cases with various layout patterns for the heat pipes assemblies. The effect of the layout of the assemblies as well as the effects of the spacing between and length of fins of the heat pipe assemblies were also examined.

Figure 3 shows the flow field in the ventilation stack without a heat-pipe assembly. The average vertical velocity at the stack outlet and the volumetric flow rate through the stack are 0.112 m/s and 0.0447 m^3/s. Because the simulation is two dimensional, the flow rate reported is for an stack with a depth of 1m. The average vertical velocity is defined as

$$V_{V-average} = \frac{\sum_{i=1}^{N} V_{V-i}}{N}$$

where $V_{V\text{-}i}$ is the vertical component of air velocity at i'th cell in the outlet of the stack and N is the total number of cells in the inlet/outlet cross-section. The volumetric flow rate through the stack is the product of the average velocity and the duct cross section. This will be compared with the flow rate when the heat pipe assemblies are inserted in the stack to obtain the insertion flow loss (IFL) as shown in the following sections.

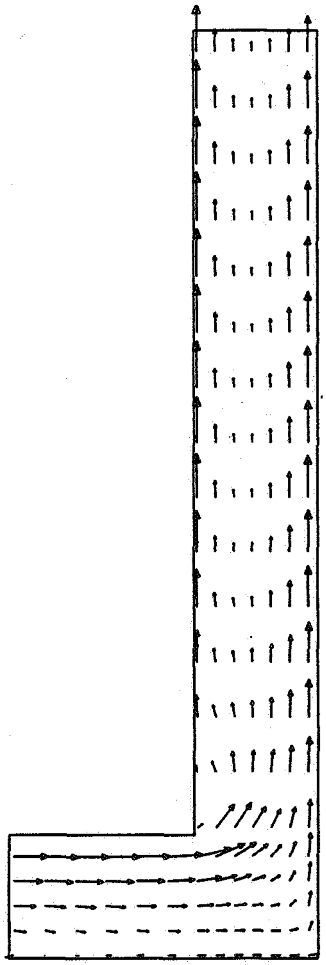

Figure 3. Flow field in the stack.

The effect of the relative location of the heat pipe assemblies was examined by computing six cases, each with a different layout pattern for the heat-pipe assemblies. , as indicated by Layout 1 - Layout 5 in Figure 4. In all six cases, the heat pipe assemblies are grouped around the centre of the vertical stack and the height of the lower horizontal surface of the bottom row of assemblies is 1.521 m. The spacing between neighbouring rows and columns of assemblies are 0.09 and 0.0 m, respectively. For Layout 1, the volumetric flow rate through the stack is 3.38 m³/s. The reduction of flow rate caused by the insertion of the heat-pipe assembly (IFL) is 26.6%. This insertion flow loss (IFL) is defined as

$$IFL = \frac{Q_{noHP} - Q_{HP}}{Q_{noHP}}$$

where Q_{noHP} and Q_{HP} are the volumetric flow rates in the stack with and without the heat-pipe assembly, respectively. It has been demonstrated that IFL is the more appropriate indicator of flow loss produced by obstacles submerged in natural convective flow such as that in a natural ventilation stack. The traditional indicator of pressure loss due to the insertion obstacles is not suitable [6].

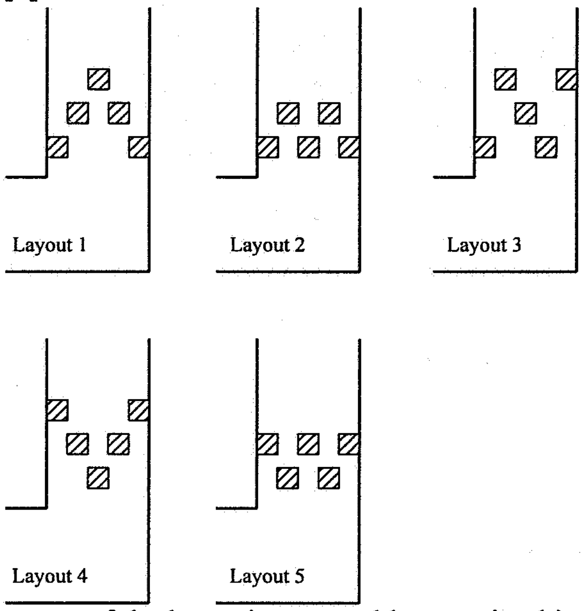

Figure 4. Layout of the heat-pipe assembly examined in this study.

The IFL's corresponding to the other four layouts of the heat-pipe assembly are listed in Table 1, together with that for Layout 1. As shown in the table the spatial arrangement of heat pipe assemblies that produced the least insertion flow loss is Layout 4 where the assemblies are placed in a pattern of an arrow facing the flow direction. Reversing the direction of the arrow in line with the flow direction lead to one of the highest flow loss found in this study. The second best arrangement producing low IFL is Layout 3 where the assemblies are located in three rows. The other two layouts (Layout 2 and 5) where the assemblies are placed in only two rows give rise to the highest insertion flow losses.

Table 1. IFL of for various layout patterns indicated in Figure 4.

	IFL (%)
Layout 1	26.6
Layout 2	33.6
Layout 3	19.0
Layout 4	8.1
Layout 5	28.9

The effect of finning arrangement on IFL was also studied by computing 6 additional cases. It is well known that reducing fin spacing and increasing fin length would increase heat transfer performance of a heat exchanger but at the cost of increasing resistance to the flow and flow loss. The extent of the flow loss was investigated by computing IFL in stacks with various numbers of uniformly spaced fins. This number was increased in stepwise fashion from 1 to 4. Although the number used is smaller than that normally found in heat exchangers, it nevertheless allows concrete conclusions to be drawn. Increasing the fin number further would require substantially more computing resources. The four-fin example is illustrated in Figure 5. The fin planes are perpendicular to the plane of the computational domain and have lengths of 0.314 m. The heat pipe linking the fins was not modelled as doing so in a 2-D simulation would block the stack. In addition, it omission is acceptable because the heat pipes are small in diameter and contributed little to flow resistance compared with the fins. The fins are located around half full-height of the vertical stack with their lower edges at 1.566 m above the floor of the horizontal stack. The stack geometry and thermal boundary conditions used are identical to those described in section 3.

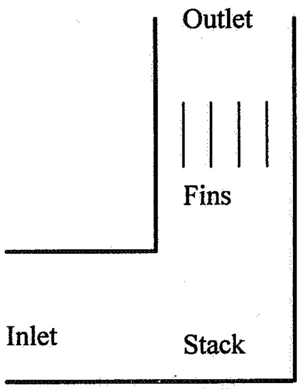

Figure 5 Schematic of arrangement of fins used in the computation.

Figure 6 shows the effect of fin spacing which is the distance between neighbouring fins. As the fin number increases from 1 to 4 the flow loss increases exponentially to over 30%. The IFL would be much greater if a larger number of fins were used as would be the case in a practical heat exchanger design. Obviously, having more than two fins in a 0.4 m wide stack would be undesirable.

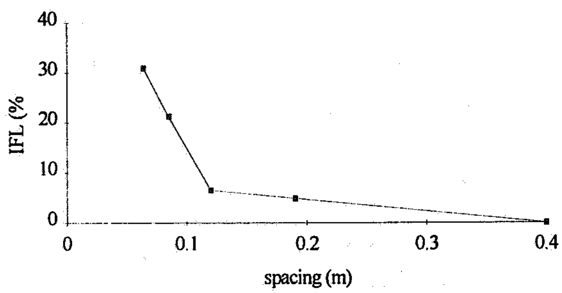

Figure 6. Effect of fin spacing on flow loss

The effect of fin length was examined by computing three cases which were identical to the three-fin case described above except their fins have different lengths. The fin lengths for the three cases are 0.179m, 0.314m and 0.493m, respectively. As shown in Figure 7, flow loss increases with fin length but the rate of increase in flow loss reduces for larger fin lengths. Therefore, for a given total surface area of fins, using fins with a greater length causes less flow loss than fins with a smaller spacing.

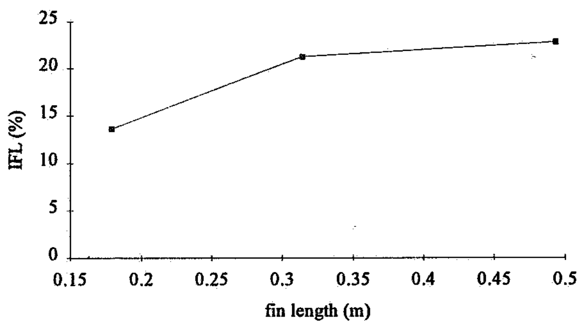

Figure 7. Effect of fin length on flow loss in the stack.

5. CONCLUSIONS

Twelve cases of natural convective flow through the stack were computed using CFD to determine the effect of the layout of heat-pipe assemblies as well as the effects of spacing and length of fins. Among the five layout patterns examined, the arrangement where the assemblies are placed in a pattern of an arrow facing the flow direction produced the least insertion flow loss. IFL was found to increase sharply with the number of fins and reached approximately 30% when only 4 fins were used. IFL also increases with fin length but the rate of increase reduces for larger fin lengths. Therefore, for a given total surface area of fins, using fins with a larger length causes less flow loss than fins with a smaller spacing.

6. ACKNOWLEDGEMENT

The authors wish to acknowledge the financial support of the European Union for this work.

7. REFERENCES

1. Bunn, R. "Learning Curve, Will natural ventilation work" Building Service, October, 1993.
2. Woolliscroft, M "The relative energy use of passive stack ventilators and extract fans" Proceedings of the 15th AIVC Conference, Buxton, UK, pp 245-256, 1994.
3. Schultz, J M and Saxhof, B. "Natural ventilation with heat recovery" Air Infiltration Review, 15(4), pp9-12, 1994.
4. Riffat, S. B., Shao, L. "Characteristics of buoyancy-driven interzonal airflow via horizontal openings" Building. Serv. Eng. Res. & Technol. 16(3), 149-152, 1995.
5. Shao, L., Walker, R. R., Woolliscroft, M. "Natural ventilation via courtyards: Application of CFD" Proc. 14th AIVC conference, pp393-403, 1993.
6. Shao, L and Riffat, S B "Flow loss caused by heat-pipes in natural-ventilation stacks", Submitted to Applied Thermal Engineering, 1996.