19th Australasian Fluid Mechanics Conference
Melbourne, Australia
8-11 December 2014

Numerical Study of Heat Pipes Effects to a 3-Dimensional Room with Driven Ventilation

Z. Abdullah[1], B.P. Huynh[1] and A. Idris[2]

[1] Faculty of Mechanical Engineering & IT
University of Technology Sydney, Sydney NSW 2006, Australia
[2] Department of Mechanical Engineering,
Universiti Kuala Lumpur, Malaysia Spanish Institute
Kulim Hi-Tech Park, 09000 Kulim, Kedah, Malaysia

Abstract

A Computational Fluid Dynamics (CFD) software package is used to investigate numerically a 3-dimensional rectangular-box room installed with heat pipes heat exchanger (HPHE). Heat pipe heat exchanger utilizing refrigerant by mean of working fluid is installed on top of a room. The air-side heat transfer and the flow pattern of a thermo-siphon heat pipes is studied with a natural driven ventilation of a building. Different opening of the inlet and outlet air where the heat pipe installed are tested with round edges opening as well as sharp edges. The standard RANS K–ε turbulence model is used. Results with different setting of heat pipe and opening characteristic, air flow rate and flow pattern as well as its temperature effects are examined.

Keywords: Computational Fluid Dynamics, heat pipes heat exchanger, natural ventilation, fluid flow, flow pattern.

Nomenclature

$U_j (j = 1 - 3)$	- Component of average velocity vector
g	- Gravity acceleration
μ_t	- Turbulent viscosity
C_μ	- Empirical constant
K	- Turbulent kinetic energy
ε	- Dissipation rate
κ	- Von Karman's constant
U_{ave}	- The average flow velocity
T_i	- Turbulence intensity
L	- Length scale

Introduction

Heat Pipes Heat Exchanger (HPHE)

This paper proposed the use of Heat Pipe Heat Exchanger (HPHE) in a room, hence analyses the air flow pattern of 100% outside air. HPHE is becoming popular as an energy saving device as there is no external power required and it has no need for maintenance. Noted by [1], 30% of the total energy consumption in the US is used by the non-domestic building and 30% of that fraction is in heating and cooling. HPHE might be one of the responses for building thermal comfort. Figure 1 show the operation and model of HPHE used. HPHE a passive, simple construction and cost effective heat exchanger is being used as a tool to demonstrate the room air flow pattern which effect by its temperature different between indoor and outdoor air. Figure 2 and 3 show the simulations series of a three pipe heat pipe heat exchanger run on ANSYS-Fluent using acetone as a working medium. The comparisons was to show different temperature between the inlet and the outlet of the heat pipes although the study of the HPHE efficiency itself is not discussed in detail in this paper. The simulations show that the delta temperature different between 2 to 3°C can be easily achieved with a row of three pipe, heat pipe heat exchanger.

HPHE is design to have high thermal conductance in steady state operation, and with only a small temperature difference, it can transfer high amount of heat. Moreover, simplicity of design and manufacturing, small temperature drops, wide temperature application range and the ability to control and transport high heat rates at various temperature levels are the unique characteristics of heat pipes [2]. The HPHE used in this modelling measured at an area of 1 m2 (2 m x 0.5m) at both inlet and outlet. According to [3] in his experiment of air heat exchangers with long heat pipes, temperature differences from 1 to 5°C were experienced when experimental with temperature differences between bath and evaporator temperatures. Analysis from Tom Brooke from Heat Pipe Technology [4] with a wrap-around heat pipes installed in an air conditioning system, found that the pre-cooled air was capable to convert a differential of 10°C sensible heat.

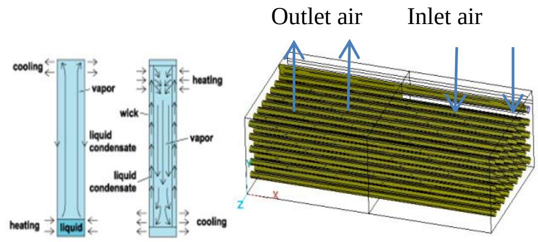


Figure 1: Heat Pipe Heat Exchanger (HPHE) model, as a heat transfer tool for cooling and heating.

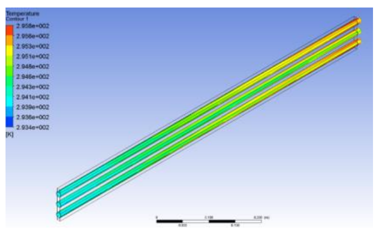


Figure 2: Heat Pipe Heat Exchanger (HPHE) simulation model run on ANSYS-Fluent using water as working medium.

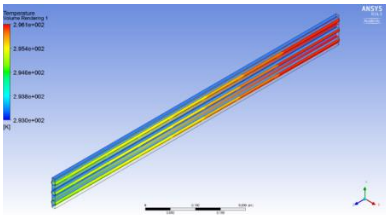


Figure 3: Heat Pipe Heat Exchanger (HPHE) simulation model run on ANSYS-Fluent using acetone as working medium.

Ventilation

The differences of air density between the inside and outside of building created natural ventilation. To achieve low-energy building and clean environment, air change is needed to be acceptable for human thermal comfort. Regarding to ASHRAE standard 55, a heat balance model of a human body is exclusively influenced by environmental factors such as temperature, thermal radiation, humidity, air speed, human activity and clothing [5]. Great attention on the design of natural ventilation to replace indoor air with fresh outdoor air without any energy consumption is being focused as it also helps to overcome health problems related to an insufficient maintenance of HVAC system [6]. Suggested by [7], [8], [9] and [10], compare to heating, ventilation and air conditioning (HVAC) system, natural ventilation is the best approach in providing comfort.

This computational study focused on driven ventilation, assumed at 3m/s, entering room dimensions of 8m x 5m x 5m, a comparison to room size suggested by [14] and [15]. As mention by [16], the air speeds induced by a temperature difference of 10°C in a building of 5 m high are of the order of 1.3 m/s and the average of the U.K. average wind speeds are in the order of 4 m/s.

Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics uses numerical methods and algorithms to solve problem involves fluid flow and predict flow behaviour based on the mathematical modelling. Model is based on fundamental governing equations of fluid dynamics namely the conservation of mass, momentum and energy. The fundamental basis of any CFD problems is the Navier-Stoke equations (conservation law). The four terms in the general differential equation are the unsteady term (transient), the convection term, the diffusion term and the source term [11]. The are many research work on natural ventilation such as [12], who study the flow structure of fluid-driven natural ventilation system using Particle Image Velocimetry (PIV) measurement and Computational Fluid Dynamics (CFD), [13] simulated natural ventilation of a large semi-enclosed stadium in an urban area using CO2 concentration decay method with CFD and [6] study on wind–driven natural ventilation in a buildings.

CFD-ACE from ESI software is used to obtain the result of the satisfactory convergence or when the steady state reached. The numerical solution of CFD-ACE processes continuity and Navier-Stokes equations work in three steps; pre-processing, solving and post-processing.

Methodology

Problem Description

The work involved with studies of two different driven ventilation patterns, namely cross and single-sided ventilation. A wind velocity of 3m/s is assumed to enter the inlet of an opening, guided by a guide vane.

Single-sided opening have both inlet and outlet at the same façade for a room with less option for ventilation. The edges for the opening of the single-sided are shape by sharp edges for Case 1. The cross ventilation have two opening for inlet and outlet, each located on top and on the side of the room respectively. The location of the inlet for Case 2 is in the centre of the roof while in Case 3 the inlet is placed near to an edge of a wall. It is to show the effect of an induced air on the roof. It will have the tendency to provide larger flow rates. It is shown in Figure 5 to 7.

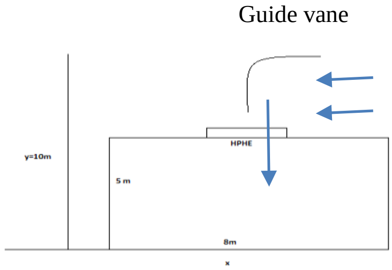


Figure 4. Wind of a velocity of 3 m/s is assumed to enter the room through inlet opening on top by guide vane.

Pre-Processing

Pre-processing helps in geometry, domain of interest and meshing. The modelling in this paper works with CFD-GEOM as the first process. Three building models were used as below;

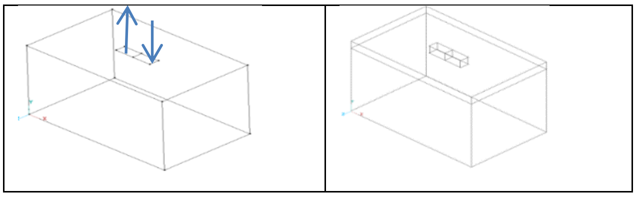


Figure 5. **Case 1.**

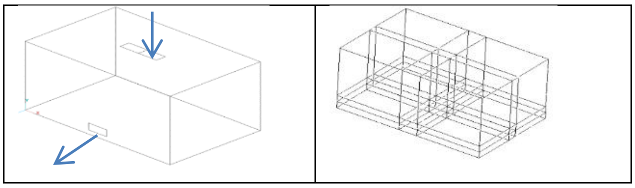


Figure 6. **Case 2**

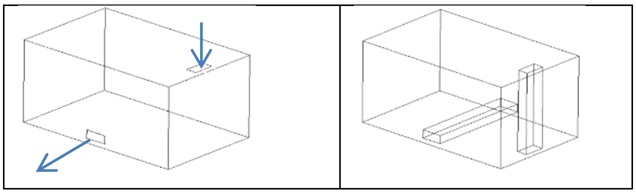


Figure 7. **Case 3**

Figure 4-7: Opening for inlet and outlet configurations for all cases. Arrows show the air down-flow directions at inlet and outlet of the room.

The mathematical model for the computational solution used turbulent flow, 3D Navier-Stokes continuity and energy equation. The governing equations are [13], [14];

$$\frac{\partial U_j}{\partial x_j} = 0 \qquad (1)$$

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{x_i} + \frac{\partial}{\partial x_j}\left[v\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \overline{u_i u_j}\right] - \beta\left(T - T_{ref}\right) \qquad (2)$$

$$\overline{-u_i u_j} = v_t\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \frac{2}{3}\mathrm{K}\delta_{ij}$$

$$\rho c\left(\frac{\partial T}{\partial t} + U_j \frac{\partial T}{\partial x_j}\right) = k\frac{\partial^2 T}{\partial x_j \partial x_j} - \rho c \frac{\partial}{\partial x_j}\left(\overline{u_j T'}\right) + \Phi + \emptyset \qquad (3)$$

$$\overline{u_j T'} = \frac{v_t}{\sigma_t}\left(\frac{\partial T}{\partial x_j}\right)$$

$$\Phi = \mu\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j}$$

$$\frac{\partial K}{\partial t} + U_j\frac{\partial K}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(v + \frac{v_t}{\sigma_K}\right)\frac{\partial K}{\partial x_j}\right] + v_t\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j} - \varepsilon \tag{4}$$

$$K = \frac{1}{2}\,\overline{U_i U_j}$$

$$\varepsilon = v\left(\frac{\partial u_i}{\partial x_j}\right)\left(\frac{\partial u_i}{\partial x_j}\right)$$

$$\frac{\partial \varepsilon}{\partial t} + U_j\frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(v + \frac{v_t}{\delta\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_1 r\frac{\varepsilon^2}{K}v_t\left[\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j}\right] - C\_1\frac{\varepsilon^2}{K} \tag{5}$$

Where subscript t refers to turbulence

$$\mu_t = {}^{\rho C_\mu K^2}/_{\varepsilon}\,;\ v_t = {}^{\mu_t}/_{\rho};$$

Cμ=0.09; C1=1.44; C2 =1.92; σK =1.0; σε =1.0; reference temperature = 300K

For the simulation, the pressure assumed to be constant, the K and ε value at inlet used;

$$K = \frac{3}{2}(U_{ave}T_i)^2 \tag{6}$$

$$\varepsilon = \frac{C_\mu{}^{3/4}K^{3/2}}{KL} \tag{7}$$

Where U_{ave} is inlet velocity, T_i is turbulence intensity (here taken to be about 2%), L is here taken to be a “reasonable length” of 1m and K = 0.41 is the Von Karman constant. Past experience has shown, however, that the values of k and ε imposed at inlet to the computational domain play only an insignificant role, as long as they are both small.

Solver

A CFD-ACE+GUI and CFD-ACE + BG solver are used to solve the governing equations that are related to the flow physic problems, based on the given material properties, flow physic model and the boundary conditions. To satisfactorily converge all properties and conditions, Finite Volume Method is employed to solve velocity components, pressure and K - ε (epsilon) scheme. Several attempts of meshing test were taken to satisfy the results. 2nd order spatial differential scheme is used. Fluid properties used for the simulations processes are corresponding to constant air at 300 K and standard pressure at sea level of 101.3 kPa, where the temperature inlet through the HPHE is assumed to be 305 K. Boussinesq approximation where the fluid density concerning the buoyancy force is affected by temperature change are assumed, with reference temperature Tref is 300K and Vol. Coef. Th. Exp. is 0.003333 1/K. Other molecular properties include ρ = 1.1614 kg/m^3, μ = 1.846 x 10^{-5} N-s/m^2, ν = 1.589 x 10^{-5} m^2/s.

Total nodes and cells that satisfied the convergence were given below for all cases;

	Total cells	Total Nodes
Case 1	324000	292668
Case 2	216000	185193
Case 3	81000	73167

Table 1: Show the total cells and nodes used for solving processes.

Post Processing

CFD-View analysed the convergence results and are presentable in a graphical presentation with different plots, streamlines and data curve. Below are the 3-dimensional graphical velocity magnitudes for air flow traces for all cases.

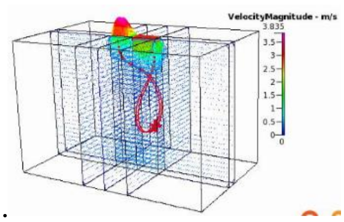


Figure 8. **Case 1**

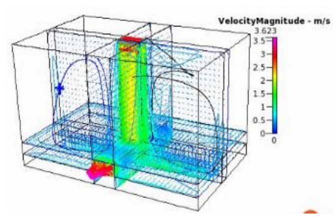


Figure 9. **Case 2**

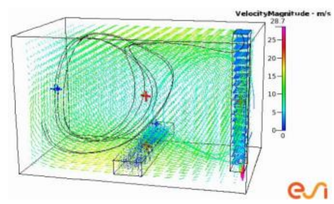


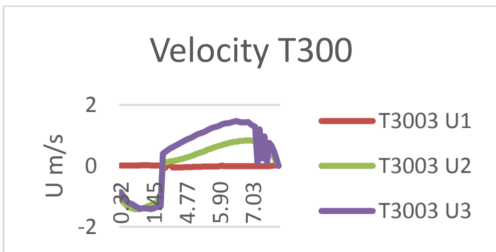
Figure 10. **Case 3.**

Figure 8-10: Show the graphical velocity magnitude in all cases

Result and Discussion

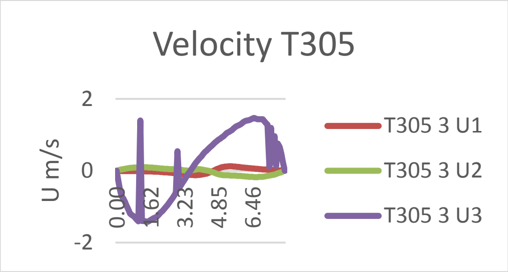
The purpose of the study is to investigate the air flow pattern and flow rate with different opening and distribution of temperature. Cases were evaluated by velocity and mass flow rate across the room.

Graph 1 shows the distribution of air velocity across the room with all cases at 300K. It shows that the velocity take-off is greater with Case 1 when entering the inlet. Half way of the room it shows that Case 3 has a larger velocity than the other cases.



Graph 1: Velocity profile for all cases at 300K.

Graph 2 shows the distribution of air velocity across the room when the temperature is increase to 305K when the air pass through the heat pipe.

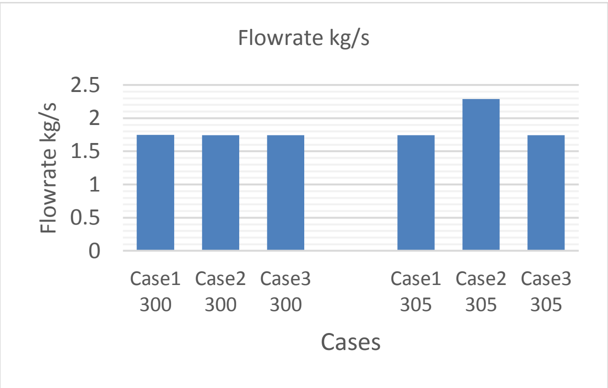


Graph 2: Velocity profile for all cases at 305K.

It is found that the highest flow rate was recorded by Case 2 which is cross ventilation with the inlet opening located at the centre of the roof top. The position is such that it allow greater flow rate to enter the room. The other configurations seems to record lower values of flow rate.

	Flow inlet at 300K, kg/s	Flow inlet at 305K, kg/s
Case 1	1.74710E+00	1.74210E+00
Case 2	1.74210E+00	2.2903E+00
Case 3	1.74210E+00	1.7424E+00

Table 2: Numerical values of flow rate in all cases at inlet of 300K and 305K.



Graph 3: Mass flow rate for all cases.

Conclusion

Heat pipe heat exchanger was the main equipment used to increase the inlet temperature. Different temperature between the inlet and the room created a flow of natural driven ventilation. A wind velocity of 3 m/s guided by a guide vane is assumed to enter the inlet of an opening and passed through a heat pipe. Simulations were performed using standard k-ε model from Reynolds Average Navier-Stokes (RANS) turbulence equation. Result of the velocity and flow rate distribution was determined according to cases of opening configurations. Case 3, a cross-ventilation where the inlet opening of a HPHE is located at the edge of the roof, shows good option for better flow rate using driven ventilation.

Further investigation involving experimental studies of HPHE to influence the air density to the room for thermal comfort is needed to validate the numerical result. Other configuration of opening for inlet and outlet, direction of driven ventilation and temperature different should be performed.

References

[1] P. F. Linden, The Fluid Mechanics of Natural Ventilation, *Annual Review of Fluid Mechanics*, 1999, **31**, 201-238.

[2] Yau, Y.H. & Ahmadzadehtalatapeh, M., A Review on the Application of Horizontal Heat Pipe Heat Exchangers in Air Conditioning Systems in the Tropics, *Applied Thermal Engineering*, 2010, **30**, 77-84.

[3] Ong, K.S. & Haider-E-Alahi, M., Performance of a R-134a-Filled Thermosyphon, *Applied Thermal Engineering*, 2003, **23**, 2373-2381.

[4] Brooke, T. *Optimizing Wrap around Heat Pipes*, Heat Pipe Technology, 2007

[5] de Dear, R.J. & Brager, G.S., Thermal Comfort in Naturally Ventilated Buildings: Revisions to ASHRAE Standard 55', *Energy and Buildings*, 2002, **34**, 549-561.

[6] Evola, G. & Popov, V., Computational Analysis of Wind Driven Natural Ventilation In Buildings, *Energy and Buildings*, 2006, **38**, 491-501.

[7] Jamaludin, A.A., Hussein, H., Mohd Ariffin, A.R., & Keumala, N., A Study on Different Natural Ventilation Approaches at a Residential College Building with the Internal Courtyard Arrangement, *Energy and Buildings*, 2014, **72**, 340-352.

[8] Nicol, J.F., & Humphreys, M.A., Adaptive Thermal Comfort and Sustainable Thermal Standards for Buildings, *Energy and Buildings*, 2002, **34**, 563-572.

[9] Su, X., Zhang, X., & Gao, J., Evaluation Method of Natural Ventilation System Based on Thermal Comfort in China, *Energy and Buildings*, 2009, **41**, 67-70.

[10] Wong, N.H., Feriadi, H., Lim, P.Y., Tham, K.W., Sekhar, C., & Cheong, K.W., Thermal Comfort Evaluation of Naturally Ventilated Public Housing in Singapore, *Building and Environment*, 2002, **37**, 1267-1277.

[11] Priyadumkol, J., & Kittichaikarn, C., Application of the Combined Air-Conditioning Systems for Energy Conservation in Data Centre, *Energy and Buildings*, 2014, **68**, Part A, 580-586.

[12] Bangalee, M.Z.I., Miau, J.J., Lin, S.Y., & Yang, J.H., Flow Visualization, PIV Measurement and CFD Calculation for Fluid-Driven Natural Cross-Ventilation in a Scale Model, *Energy and Buildings*, 2013, **66**, 306-314.

[13] van Hooff, T., & Blocken, B., CFD Evaluation of Natural Ventilation of Indoor Environments by the Concentration Decay Method: CO2 Gas Dispersion from a Semi-Enclosed Stadium, *Building and Environment*, 2013, **61**, 1-17.

[14] Kivva T., Huynh, B. P., Gaston, M. & Munn, D., A Numerical Study of Ventilation Flow Through a 3 Dimensional Room with a Fan, *Turbulence*, *Heat and Mass Transfer 6*, Hanjalić, K., Nagano, Y. & Jakirlić, S., (editors), Begell House, Inc, 2009

[15] Idris, A., Computational Study of Single-Sided Ventilation through a 3-Dimensional Room with Rounded Edges, *American Society of Mechanical Engineer*, 2013, **9**.

[16] Hunt, G.R., & Linden, P.P., The Fluid Mechanics of Natural Ventilation, Displacement Ventilation by Buoyancy-Driven Flows Assisted by Wind, *Building and Environment*, 1999, **34**, 707-720.