Application of CFD for Air Flow Analysis underneath Natural Ventilation with Forced Convection in Roof Attic

C. Nutphuang, S. Chirarattananon, and V.D. Hien

Abstract—In research on natural ventilation, and passive cooling with forced convection, is essential to know how heat flows in a solid object and the pattern of temperature distribution on their surfaces, and eventually how air flows through and convects heat from the surfaces of steel under roof. This paper presents some results from running the computational fluid dynamic program (CFD) by comparison between natural ventilation and forced convection within roof attic that is received directly from solar radiation. The CFD program for modeling air flow inside roof attic has been modified to allow as two cases. First case, the analysis under natural ventilation, is closed area in roof attic and second case, the analysis under forced convection, is opened area in roof attic. These extend of all cases to available predictions of variations such as temperature, pressure, and mass flow rate distributions in each case within roof attic. The comparison shows that this CFD program is an effective model for predicting air flow of temperature and heat transfer coefficient distribution within roof attic. The result shows that forced convection can help to reduce heat transfer through roof attic and an around area of steel core has temperature inner zone lower than natural ventilation type. The different temperature on the steel core of roof attic of two cases was 10-15 $^{\circ}$K.

Keywords—CFD program, natural ventilation, forced convection, heat transfer, air flow.

I. INTRODUCTION

THE potentials investigation of the roof attic improves natural ventilation, using computational fluid dynamics (CFD) three-dimensional modelling. In many cases, computer programs are written to calculate heat flow across walls and windows into buildings. Typical computer simulation programs can calculate cooling load and energy consumption of air-conditioning system in buildings [1]-[5].

C. Nutphuang is Joint Graduate School of Energy and Environment, King Mongkut's University of Technology Thonburi, 126 Prachauthit Road,Tungkru,Bangkok,Thailand,10140.(phone:+6624708626;fax:+6647086 26;email:jensuek@hotmail.com).

S. Chirarattananon is Joint Graduate School of Energy and Environment, King Mongkut's University of Technology Thonburi, 126 PrachauthitRoad,Tungkru,Bangkok,Thailand,10140.(phone:+ 662872 9014-5 ext. 4128;fax:+ 662872 9017;email: surapong@jgsee.kmutt.ac.th).

V.D. Hien is Joint Graduate School of Energy and Environment, King Mongkut's University of Technology Thonburi, 126 PrachauthitRoad,Tungkru,Bangkok,Thailand,10140.(phone:+ 662872 9014-5 ext. 4128;fax:+ 662872 9017;email: hienv278@yahoo.com).

This case involves heat transfer and heat flows in three dimensions. The steel structure supports roofs in the zone of a roof attic. The parts of the structure that touches the roofs are at higher temperature and transfer heat by conduction to other parts of the structure in the zone. The air surrounding this structure also receives convective heat and its temperature should differ from that of the air in the rest of the zone [5]-[8]. A simulation program under the condition of air in a zone is homogeneous. When there are sufficient differences of air temperatures in different locations, air will flow. The existence of solid object and three dimensional heat transfers that follow render conventional simulation programs inapplicable [9]-[10]. CFD models are capable of predicting the air flow patterns and distributions of temperature on steel core arising from point or area source at all positions over a finely-spaced grid filling the total volume of a roof. CFD models have been used to predict general roof airflow, inlet/outlet arrangement, occupant effects, displacement ventilation, and contaminant transport. But their application requires much specialized knowledge and fluid engineering judgment. Some significant difficulties with CFD, especially when it is necessary to use three-dimensional analysis, include: that setting up the model, identifying, and specifying appropriate boundary conditions are difficult and time consuming that the calculations are time consuming (especially for industrial buildings); that large amounts of computer memory are required (because of the restrictions on grid size) to enable adequate treatment of turbulence; and that they sometime fail to converge to a solution. Furthermore, the large volume of the output requires considerable effort in post-processing and visualization in order to understand the results. CFD representation of dynamic boundary conditions (such as ventilation, infiltration rates. surface temperatures) was deficient [11]. This lack of proper information at the boundary may lead to the reduction of flow modelling accuracy. An analytical procedure was to reduce heat transmission through a roof that the result of the researcher has made a difference for heat transfer through the roof with radiation, convection and conduction within a heat insulating under wall and roof structure. The resulting expressions for heat and temperature were coupled with convection and conduction equations to form a set of heat balance equations for each heat transfer surface. They illustrated calculations

conducted to estimate heat transfer rates through a heat insulating roof structure through steel core. The result of the researcher has shown a different temperature, heat transfer coefficient, mass flow rate, and different temperature of steel core through roof attic.

II. THE ROOF ATTIC DESIGN

The roof attic designs by using solid works program make roof attic. The roof attic is divided following in the part of touching with wall, insulation, ceiling and solid object follows as color by having roof angle as equal as 30 degrees Fig. 1 by brown color is solid object, and green or yellow color are enclosed insulation. Roof attic model is 4 mx4 m dimensions by divided roof zones and isosceles triangle (gable) of north and south wall is a 1.5 m high from ceiling. Both east and west of side wall are 0.3 m high from ceiling. The ventilator is the middle of north and south façade. It is 0.5mx0.5m dimensions and inside consists of steel core, is 2mx 1.17m dimensions, and 0.1m thickness. By sides two wing of roof have 2.2mx4m dimensions. For the characteristic exterior of roof structure has some methodology for defining boundary conditions to give with roof structure in each side which it consists of north and south facade, east and west wall, ventilator, and both roof wing.
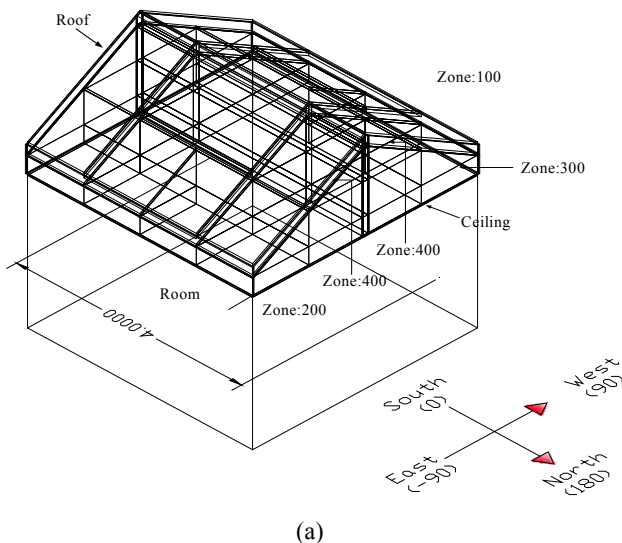
(a)
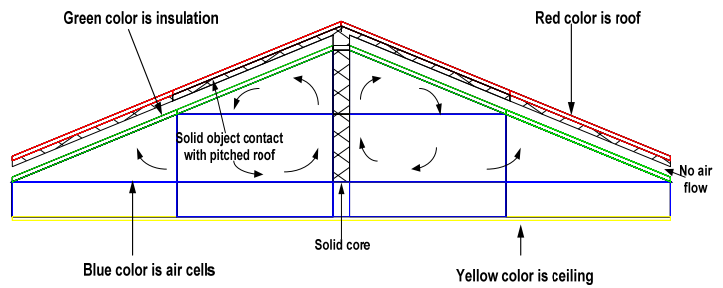
(b)

Fig.1 (a) Wireframe roof attic model 4 mx4 m dimensions by divided zones roof. (b) Solid object contacts with roof and no air flow upper insulator.

Fig. 1 presents the position of air cells each i cells(Number of air cells) in the local coordinate x_iy_i and main coordinates XYZ by looked to Z axis is constant layer(Top view) and it divided following by steel structure of that layers. We input configuration data; first, the user must define all components of cells. The components of the roof attic are air cells characteristic, walls, insulations, steel, ceiling, floor, etc. Later, steps of definition for components of the roof attic each layers are as in the followings. 1. Define the main coordinate: XYZ require OZ is horizontal. 2. Define non air-conditioning zones: 100,200,300, and 400 3.Define the ambient zone as 100.4.Define names of sections of wall, air cells in roof attic, and steel: Integer numbers.5.Define local coordinates of sections of wall and facade, air cells in roof attic, and steel. 6. Define subcoordinates of apexes of sections of wall and facade, air cells in roof attic, and steel in local coordinates. All conditions of roof are defined for using data for novel simulation program [12] by comparison with CDF between advantages and disadvantage each program.

A. Boundary and Initial Conditions for Roof Attic

To solve the continuity and momentum equations, appropriate boundary conditions were specified at all boundaries. The recorded experimental data was for 13 hours. CFD simulation run time of 13 hours that had high solar radiation which was an unsteady state process was impossible due to high number of time steps and computer time limitations. It was assumed that in an all times period of daytime, the received solar radiation by the sun as well as temperatures distribution of the roof were almost constant. Hence an overall process was modeled as 12 stages of 1 hour interval for daytime in a quasi steady state condition. Boundary conditions for wall and façade in closed area with opened area, steel and insulator were considered as constant temperature boundary. Fig. 2 showed closed area and opened area of roof attic of temperatures based on the experimental

data. The experiments were carried out in summer days of year 2006 from 8:30 A.M. till 23:30 P.M. In every 1 hour time interval, an average temperature was set as boundary condition. The steel core temperature was be specified the highest temperature and ceiling was defined the lowest temperature in roof zone. Bottom of roof and heat reservoir temperatures were almost in roof attic. Solar intensity was based upon absorption factor and emissivity of all sections in roof. For drop formation on the roof, adhesion forces were taken into account in simulations. All sections (façade, wall, steel, and ceiling) were assumed adiabatic. A no-slip wall boundary condition was specified for the air phase and free-slip boundary condition was used for the gas phase.
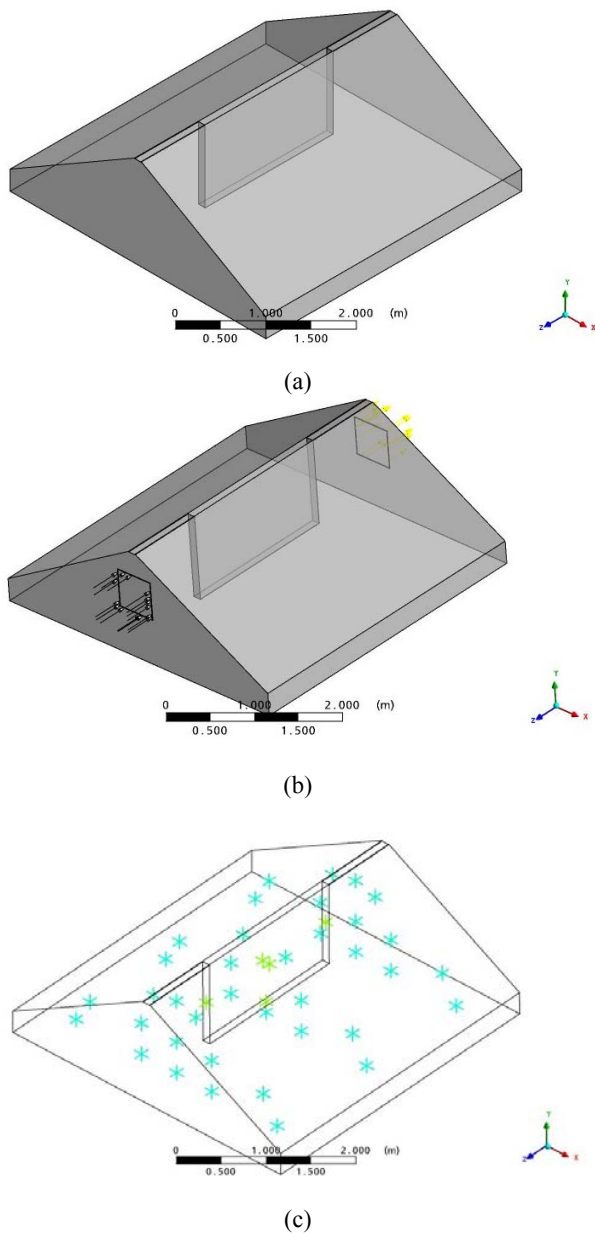

(a)

(b)

(c)

Fig. 2 (a) Closed area no vent (b) Opened square area for inlet and outlet air. (c) The 37 point of temperature for examining in CFD

Fig.2 (a) is closed area no vent and façade of north and south is z axis for first case. The wall of roof is as x- axis of east and west. Fig.2 (b) is opened area, has vent for air jet 3 m/s velocity into roof vent. Air is flowed contact within steel core and out to outlet for second case. Fig.2 (c) shows 37 point of temperature and pressure for finding each layer of roof attic. We found that calculation heat transfer three dimensions in roof attic of CFD. They consist of mainly three parts which are 1.radiation 2.conduction and 3.convection. In the part of conduction and convection are calculated in three dimensions by referenced inner boundary conditions.

III. MECHANISMS OF AIR FLOW

When a wall or roof within zone is poorly insulted or wall surface exposed to solar radiation, the surface temperature is different from the surroundings and there is free convection between the wall surface and the surrounding air. In case of driving forces is jet or having forced convection with fluid moving at higher velocity than surrounding fluid. Therefore, it can be shown as source in jet system.

A. Natural Convection

Natural(or buoyancy-driven) convection occurs that when a fluid comes into contact with a heat surface, heat transfer takes place by conduction and fluid temperature variations are established which give rise to density variations. Buoyancy forces then establish fluid motion to carry away the conducted heat.

$$Nu_n = kf(\mathrm{Pr}, Gr) \qquad (1)$$

Where Nu_n is the Nusselt number for natural convection, Pr is the Prandtl number, Gr is the Grashof number, k is a constant and f is a function.

Heat ceiling ($9 \times 10^8 < Gr < 1 \times 10^{11}$)

$$h_{cn} = \frac{0.704}{D^{0.601}}(\Delta T)^{0.133} \qquad (2)$$

$$Nu_n = 1.78(Gr)^{0.133} \qquad (3)$$

B. Forced Convection

Forced convection is concerned with the transfer of heat between a moving fluid and a solid surface where the fluid motion is caused by external means. When fluid motion is caused by some external forces such as fan, jet or wind power, convection is termed forced.

$$Nu_f = kf(\mathrm{Pr}, \mathrm{Re}_f) \qquad (4)$$

Where Re_f is the Reynolds number, k is the constant of various flow configurations and f is a function whose values are found from experiments. Nusselt number correlations usually scale the nusselt number to a power n of the reynolds number in the form:

$$Nu_f \approx (\mathrm{Re})^n f(\mathrm{Pr}) \tag{5}$$

For air, this expression reduces to:

$$Nu_f \approx b(\mathrm{Re})^n \tag{6}$$

Eventually, defined area represents a portion of a partition, convective exchange takes place which can be modeled by heat coefficient h_{cv}, The heat flow then reads.

$$q_{cv} = h_{cv} A (T - T_0) \tag{7}$$

Where T is the temperature of outside air

IV. AIR FLOWS ANALYSIS

A. Mass Flow Rate

$$\text{Rate in- Rate out} = \sum_{j=0}^{N_z} \sum_{l=1}^{N_l} m_{jil} \tag{8}$$

Where $m_{jil} \geq 0$ is the air mass flow rate of flow path l leaving sub-zone i. N_z is the total number of the zones (or sub-zones) within the solution domain. N_l is the total number of flow paths between sub-zones j and i

$$\sum_{j=0}^{N_z} \sum_{l=1}^{N_l} m_{jil} + m_{si} = 0 \tag{9}$$

Where m_{si} is air flow source in sub-zone i, $m_{si} > 0$ if it is a source, and $m_{si} < 0$ if it is a sink.

B. Pressure Constrain

$$dm_{j,i} = C_d \rho (\Delta p_{j,i})^n dA \tag{10}$$

The differential mass flow $dm_{j,i}$ through differential area dA (the adjoining face area) is expressed and, for air flow from j to i, the mass flow rate $m_{j,i}$ through the cross-sectional area A is,

$$m_{j,i} = \int dm_{j,i} = C_d \rho (\Delta p_{j,i})^n A \tag{11}$$

$$\Delta p_{j,i} = p_j - p_i \tag{12}$$

For air flow through a vertical interface:

$$\Delta p_{j,i} = (p_j - p_i) - \frac{1}{2}(\rho_i g h_i + \rho_j g h_j) \tag{13}$$

For air flow through a horizontal interface

Where h_i and h_j are the height of the sub-zones i and j respectively ρ_i and ρ_j are the densities of sub-zones i and j respectively and g the gravitational acceleration.

C. Heat and Temperature

The set of mass balance equations above are not complete. Air density in each zone is a function of dryT-bulb temperature of air. These are a need for a set of energy balance equation. Air transfer heat with the surfaces of solid objects it comes into contact through convection and conduction.

$$\text{Rate in} - \text{Rate out} = \sum_{j=0}^{j=N_z} \sum_{l=0}^{l=N_l} m_{jil} c_p (T_j - T_i) + q_{convi} \tag{14}$$

$$\text{Rate in} - \text{Rate out} = \sum_{j=0}^{j=N_z} \sum_{l=0}^{l=N_l} m_{jil} c_p (T_j - T_i) + q_{convi} \tag{15}$$

$$q_{conduction} = \sum_k \frac{d_{ik}}{k_a} A_{ik} (T_{ik} - T_i) \tag{16}$$

The rate of air transferred into and out of a subzone is given as rate of heat into-rate heat out of a subzone

Where h_{ik} is the convection heat transfer coefficient between surface k in zone i, d_{ik} is the distance between the centroid of subzone i and surface k

$$\sum_j \sum_l m_{jil} C_p (T_j - T_i) +$$
$$\sum_k (h_{ik} + \frac{d_{ik}}{k_a}) A_{ik} (T_{ik} - T_i) + q_{si} + m_{si} C_p T_{si} = 0 \tag{17}$$

If there are heats source q_{si} and heat sink $m_{si} C_p T_{si}$ in the accounted in the mass balance equation. The overall equation of heat transfer is in a subzone i.

V. RESULTS AND DISCUSSIONS

From the design of roof attic under natural ventilation, and force convection in three dimensions was to define value in calculation, to define center of mass each air cell, and boundary conditions for roof attic. Air cells were divided to following solid object in tetrahedron shapes. For numerical solution of turbulent flow, a high-Reynolds-number k-ε model was suitable.

Computational domain sizes were selected so as to fulfill the requirements of free-stream conditions whilst ensuring that grid geometrical characteristics satisfy the physical limitations of the standard k-ε model.
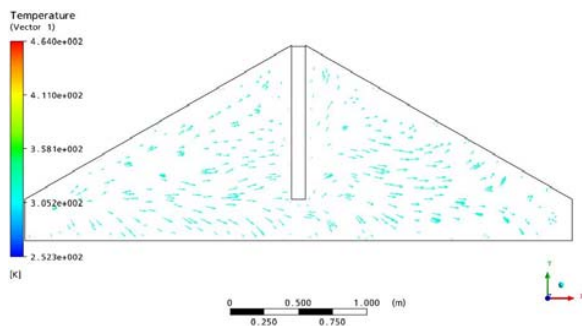
Fig. 3 Temperature flux flow of air in roof attic

Fig. 3 for the temperature flux diving force has to move from east roof to west roof in 13 hours because west roof appears heat transfer higher than west roof, which it has to accumulate of high temperature this area.
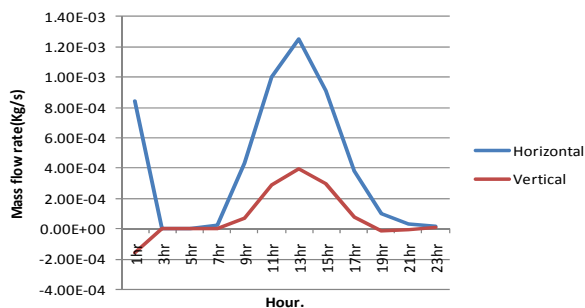
Fig. 4 Mass flow rate of air flow in the same layers

Fig. 4 shows mass flow rate of air in horizontal and vertical that have a difference movement of air. The horizontal has mass flow rate higher than in vertical at 13 hours on same level of air layers of daytime.
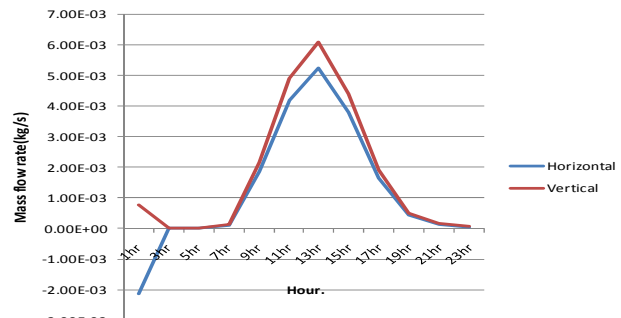

Fig. 5 Mass flow rate of air flow in the different layers

Fig. 5 shows mass flow rate of air in horizontal and vertical that have a difference movement of air. The vertical has mass flow rate higher than in horizontal at 13 hours on different level of air layers of daytime.
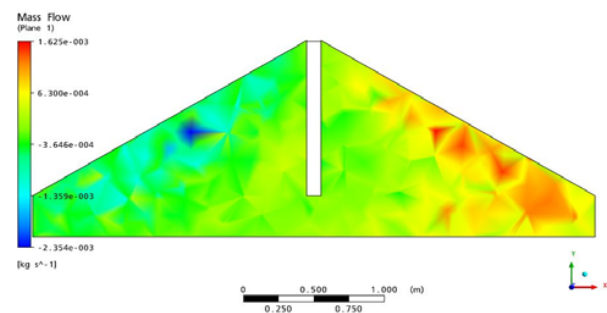

Fig.6 Mass flow of air distributes no vent in roof attic

Fig. 6 for the west area of roof attic (red area) in 13 hours has the scattering mass flow rate higher than the east area (blue area) because west area has been gotten solar radiation in afternoon. The Reynolds number was less than 10^5
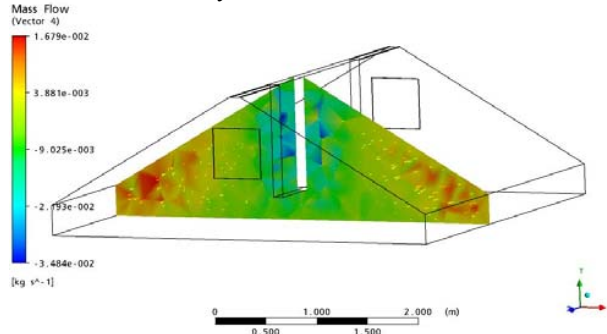

Fig.7 Mass flow of air distributes to have vent in roof attic

Fig. 7 for mass flow has distributed both side of roof wing by wind force from vent 3 m/s velocity. The east and west of roof edge has accumulation of mass more than steel core area around. The temperature in daytime is as 30-40 ^{o}C by force convection. In the part of night time core steel temperature has 20-30 ^{o}C which it has value higher than couple wings edge.
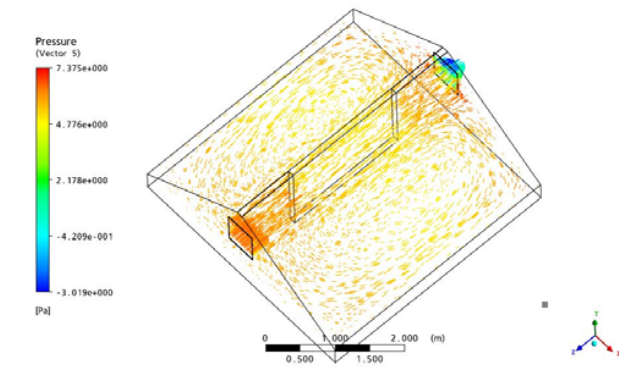
Fig. 8 Pressure flux flow of air to have in roof attic

Fig. 8 for the pressure flux diving force has to move around in roof, which has turbulence flow inner roof attic. The Reynolds number has more than 10 6. Air in roof has eddy flow between east and west roof edge and has pressure distribution inner roof attic.
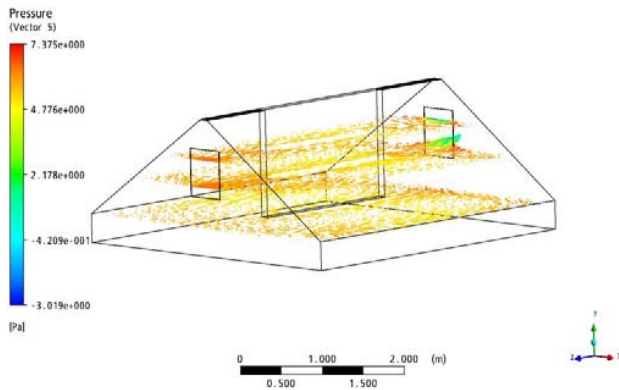

Fig. 9 Pressure flux layers flow of air to have vent in roof attic

Fig. 9 for the pressure of the lowest layer has pressure lower than upper layer, and top layer has the lowest pressure of every layer. A temperature has reversed with pressure and each layer has the different eddy and turbulence in roof attic.
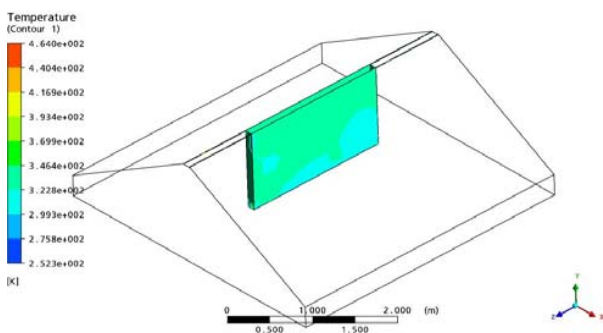

Fig. 10 Temperature distribution on the steel core no vent in roof attic

Fig. 10 shows temperature distribution on the steel core. The temperature distribution is mostly 346-322 ^{o}K at steel core area and top of steel core has temperature higher than its

bottom. In air cells near with steel core have heat transfer by convection to another air cells and conduction heat transfer from top through bottom of steel core.
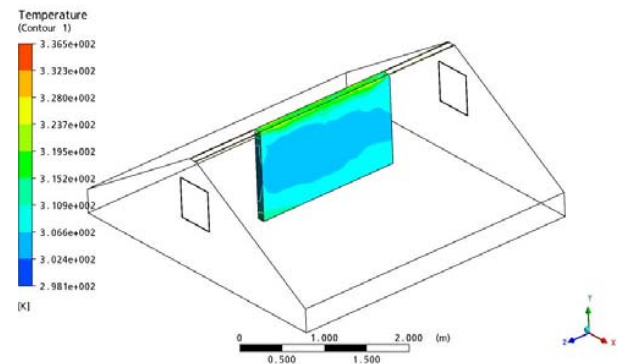

Fig. 11 Temperature distribution on the steel core has vent in roof attic

Fig. 11 shows temperature distribution on the steel core to each layer. The temperature distribution follows of steel core area and top of steel core has higher than its bottom (323.7 ^{o}K). In air cells near with steel core has heat transfer by convection to another air cell and conduction heat transfer from top through bottom of steel core as same as Fig. 10
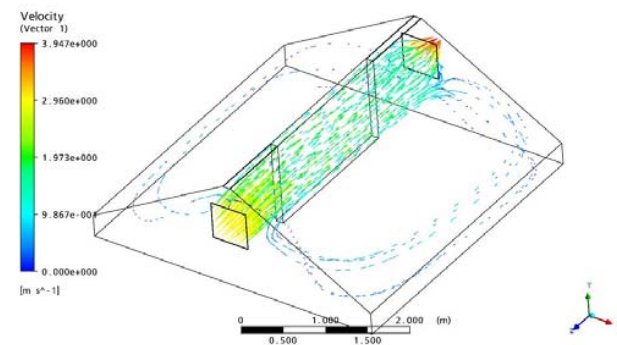

Fig. 12 Velocity of air flow 3 m/s inlet to outlet

Fig. 12 shows velocity of eddy and turbulence flow in roof attic .Inlet has 3m/s velocity of air flow and outlet has been zero pressure. The air jet in the north of façade flows to the south of outlet. The effect from air jet has increased heat transfer coefficient. If we reduced injet air from 3m/s to 2 m/s, and 1 m/s, it would have been reduced heat transfer coefficient.
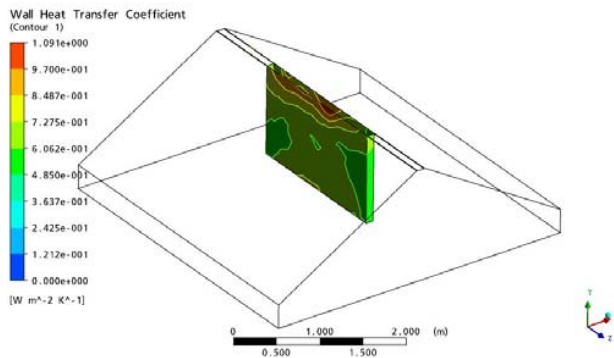
Fig. 13 Wall heat transfer coefficient on the steel core contour no vent in roof attic

Fig. 13 shows wall heat transfer coefficient on the steel core contour accumulative on the top steel and it has distributed on the middle and on the bottom area that it has been 0.363-1.019 at 13 hours in summer, 2006
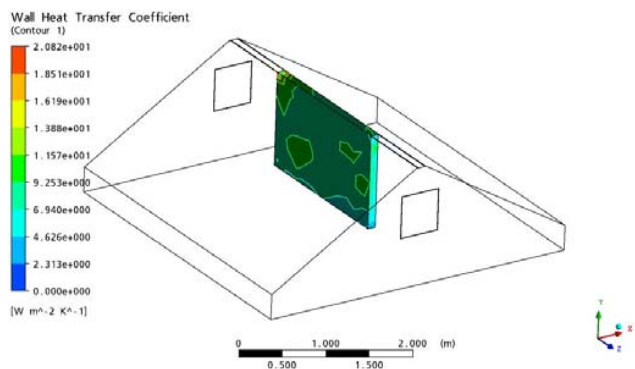

Fig. 14 Wall heat transfer coefficient on the steel core contour has vent in roof attic

Fig. 14 shows wall heat transfer coefficient on the steel core contour accumulative on the top steel core and it has distributed more good than Fig.13 on the middle and on the bottom area. It has been 2.31-20.82 at 13 hours in summer, 2006

VI. CONCLUSION

Since, the air in roof attic was divided in each layer. The CFD simulation found that the upper layer had temperatures higher than lower layer and pressures revered with temperatures for natural convection. The result of different pressure had mass of moving air and mass flow rate had mostly distribution on both wings of roof. Although, the CFD simulation showed mass flow rate of air between cells in the center of mass, showed air flow in the horizontal and vertical direction, and showed heat flux contour on the steel core for analysis how it could move in any directions of heat flux on the steel core. Besides, the design natural ventilation and forced convection system in roof and building had using CFD simulation to support for studying their air flow pattern. The results of simulation for forced convective analysis in the roof

attic were air jet in 0 o direction on its vent for turbulent flow. Air attacked directly with steel core for investigative heat transfer on the steel core and the result of temperature could well reduce heat transfer through roof tile. The temperature of forced convection on the steel core had obviously different temperature when it was compared with natural convection, was about 10- 15 ^{o}K. The forced convection helped to reduce heat transfer through roof. If air jet had more than 3m/s of velocity, roof vent would be proper high of buildings from ground, which they should have higher than 20 m up or more than that. But for forced convection in real natural, it could not predict correctly. Therefore, we might use air jet or sucked air system for ventilation in roof attic. In the studying the wind driven ventilation is processed the part of such as roof and building. Since the pressure and temperature distribution around a building are the driving force for the ventilation process inside a naturally ventilated building, the presented results are a good indication that the ventilation process can also be modelled accurately for both roof and building. This gives confidence for future research where CFD can be a useful tool for inexpensive computer experiments on the internal environment of large commercial buildings. The resulting CFD model can be used to conduct parametric studies and be of use in guiding decisions about improved design and operation changes that could lead to improved building control.

REFERENCES

[1] Cezar O.R. Negriio., "Integration of computational fluid dynamics with building thermal and mass flow simulation", *Energy and Buildings*,1998;27: 155-165.

[2] Feustel Helmut E., "COMIS-an international multizone air-flow and contaminant transport mode", *Energy and Buildings*, 30(1999):3-18..

[3] Helmut E. Feustel., "COMIS—an international multizone air-flow and contaminant transport model",*Energy and Buildings* ,30(1999):3-18.

[4] Ren Zhen, Stewart John., "Simulating air flow and temperature distribution inside buildings using a modified version of COMIS with sub-zonal divisions", *Energy and Buildings*, 2003;35:257-271

[5] Zhengen Ren, John, "Stewart.Simulating air flow and temperature distribution inside buildings using a modified version of COMIS with sub-zonal division", *Energy and Buildings* ,35,(2003):257-271.

[6] Christian Inard , Hassan Bouia , Pascal Dalicieux., "Prediction of air temperature distribution in buildings with a zonal model", *Energy and Buildings* 1996;24:125-132.

[7] John Stewart, Zhengen Ren., "COwZ—A subzonal indoor airflow, temperature and contaminant dispersion model", *Building and Environment*, 41, (2006): 1631–1648.

[8] Meir Teitel and Josef Tanny., "Heat Fluxes and Airflow Patterns T hrough Roof Windows in a Naturally Ventilated Enclosure", *Flow, Turbulence and Combustion* ,2005;74:21-47.

[9] L. Susanti a, H. Hommab, H. Matsumoto b, Y. Suzuki c, M. Shimizu. "A laboratory experiment on natural ventilation through a roof cavity for reduction of solar heat gain", *Energy and Buildings*, 2008;40:2196-2206.

[10] Sunwoo Lee, Sang Hoon Park, Myong Souk Yeo, Kwang Woo Kim. "An experimental study on airflow in the cavity of a ventilated roof",

Building and Environment ,2009;44:1431-1439.

[11] Omar S. Asfour, Mohamed B. Gadi.," Using CFD to investigate ventilation characteristics of vaults as wind-inducing devices in buildings", *Applied Energy* 85 (2008):1126–1140.

[12] Vu Duc Hien and Surapong Chirarattananon,(2006). Building Energy Simulation Program-BESim. "A presentation published in the Proceedings of the training workshop on Daylight under Tropical Sky", 17-19 May 2006, *AIT Conference Center*, Asian Institute of Technology(AIT), Bangkok, Thailand.