Numerical analysis of mixed mode convection in 3D cavity: Application to air refreshment in buildings

A. RAHMAN

M.Sc in Computational Engineering (Mécanique Numérique et Ingénierie)
Université de Strasbourg, Strasbourg 67000, France
asfik.ae@gmail.com

Abstract:

A domain with classical dimension has been studied numerically by STARCCM+ based on recent research and innovation in the field of Energy and Buildings. Conceiving temperature range of summer (May-July) in southern France, Mixed mode (Mechanical ventilation through inlet, Natural ventilation in room) ventilation method has considered as a mean of air refreshment in a classical room. Moreover, numerical analysis has performed at Reynolds Number (Re) 8.6 x 10^2, 1.7 x 10^3 to examine effects of inlet angle (horizontal, inclined) and domain wall thickness in air refreshment. Finally, all the parameters have compared by bulk temperature on horizontal planes and heat transferred from domains to find out the optimum solution.

Keywords: Air refreshment in Buildings, Mixed mode ventilation, 3D unsteady numerical analysis

Nomenclature:

Variables

m	Mass
$\dot{Q}$	Heat transfer rate
k	Turbulent kinetic energy
F_s	Factor of safety
p	Order of accuracy
r	Grid refinement factor
t	Time
$\vec{v}$	Fluid velocity vector $\vec{v}=(u,v,w)$
x	Horizontal coordinate
y	Vertical coordinate
z	Depth coordinate
C_d	Coefficient of discharge

Variables

E	Total energy
G	Production of turbulent kinetic energy
K_{eff}	Effective conductivity
N	Number of cells
P	Pressure
T	Temperature
$\dot{m}$	Mass flow rate
c_p	Specific heat
$\vec{v}'$	Velocity field correction
p'	Pressure correction
k_{wall}	Thermal conductivity
Q_{cond}	Conductive heat transfer

Greek Letters

β	Thermal expansion coefficient
ε	Turbulent rate of dissipation
ρ	Density of fluid
μ	Viscosity
$\bar{\bar{\tau}}$	Fluid tensor

Subscripts

1	Fine grid
2	Medium grid
3	Coarse grid
in	Into the room
out	Out of the room

I. INTRODUCTION

Though the idea of innovative house flourished in 17th century, the actual application started in 20th century. Many researches have proved that world is heading to an energy crisis in near future. Utilization of natural resources could be a probable solution for a greener, better and economical world.

As a consequence of energy crisis, *Thermal Regulation (RT)* has been embodied for building construction before few decades. The first thermal regulation dates back to 1974 *(RT 1974)*. Afterword, new thermal regulation goal had been set and fulfilled successively (i.e. *RT 1988, RT 2000, RT 2005, RT 2012 and RT 2000)*. The *RT 2012* was aimed new homes to consume up to 50 kWhep par m^2 par years. However, this value changes notably depending on the region and the altitude at which the house is located. This maximum consumption defines the building low consumption. Finally, *RT 2020* has also been regulated with a goal of positive energy production system in buildings which means the energy production will be more than requirement. The imminent computational work of this scientific article was a part of an innovative air refreshment system for buildings with an endeavor to attain *RT 2020* in near future.

Meanwhile, many ingenious inventions such as Free or Natural cooling, solar heating, Mechanical or Forced ventilation, mixed mode ventilation have been introduced to the world as a consequence of regulation of building construction. As a part of study, literature review has been done to explore previous research on ventilation process. For instance, ***Jiang et al.*** [1] presented an experimental and numerical computational study (LES, large eddy simulation) of natural ventilation, driven only by wind forces for simple geometries representing cross-ventilation and single-sided ventilation configurations. In order to reduce the computational cost, ***Evola and Popov*** [2] applied Reynolds Averaged Navier-Stokes (RANS) models, the standard and the RNG $\boldsymbol{K-\varepsilon}$ models, for the experimental set up described in reference [1], and also obtained reasonable accuracy for the velocity distribution inside the building model and also for the ventilation flow rates. Another study has been done by ***Mingang Jin et al***. [3] for solving natural convection problem. The study investigated Fast Fluid Dynamics (FFD) performance for simulating natural ventilation. FFD was applied to different types of natural ventilation, such as wind-driven, single-sided natural ventilation, wind-driven cross natural ventilation, and buoyancy-driven, single-sided natural ventilation. They also examined the impact of wind direction and applied FFD to natural ventilation in a building complex.

T. Catalina et al. [4] has studied a full-scale experimental campaign and a computational fluid dynamics (CFD) study of a radiant cooling ceiling installed in a test room, under controlled conditions. Their research aims to use the results obtained from the two studies to analyze the indoor thermal comfort using the predicted mean vote (PMV). During the whole experimental tests the indoor humidity was kept at a level where the condensation risk was minimized and no condensation was detected on the chilled surface of the ceiling

N.C Markatos et al [5] have considered natural ventilation as solution of increasing energy consumption due to mechanical ventilation, atmospheric pollution and global warming. They have suggested that the proper design of natural ventilation must be based on detailed understanding of airflow within enclosed spaces, governed by pressure differences due to wind and buoyancy forces.

Now, If we focus on wall thickness of the domain, an unsteady three-dimensional conjugate heat and mass transfer in an enclosure having finite thickness heat-conducting walls has been analyzed numerically by ***Sheremet et al [6].*** They have solved the governing unsteady, three-dimensional flow, energy and contaminant transport equations for the gas cavity and unsteady heat conduction equation for solid walls by using an iterative implicit finite-difference method. To take the previous study one step forward, Conjugate natural and forced convection heat transfers in a domestic model room of finite-thickness walls and a heat source have been numerically studied by ***Horikiri et al [7]***

From the analysis of previous literature relevant to natural and mechanical ventilation, it is easily apprehensible that only a few studies have been done before regarding inlet angle effect on heat removal from domain. Therefore, we will try to take old studies one step further by considering two different inlet angles which will be tested numerically for same domain in this imminent study.

During the study, a classical empty room was assumed as domain with horizontal and inclined inlet for summer season and the temperature range had been considered for southern region of France. Then the effect of inlet angle for air refreshment has been observed by numerical simulation at Reynolds number 8.6×10^2 *and* 1.7×10^3 to acquire optimal solution for the system. Meanwhile, Mixed mode (Mechanical ventilation through inlet, Natural ventilation in room) ventilation method had considered as principle of heat removal from room. As the Natural ventilation is a major part of Mixed-mode ventilation and optimally operate in small temperature difference, the system will function 30 minutes just in night when the outdoor temperature is lower than indoor. A mechanical fan was placed for forced ventilation at the inlet which located below the window and the outlet is placed just upper portion of window. The domains have been simulated with maximum critical conditions so that we can ensure about the system's highest ability regarding heat removal. Hence, window remained closed because sometime it's might not possible to open window due to dust, noise or insects especially room located in ground floor. Moreover, the door was also considered closed and there was no air pass except outlet.

II. GOVERNING EQUATION

As interaction between Natural and Forced convection has been investigated inside a three dimensional enclosure, the governing equation contains conservation of Momentum, Mass and Energy equations. The conservation of Momentum is commonly referred as the *Navier-Stokes Equation*. Additionally, conservation of Mass and Energy are also used to solve velocity and temperature field in domain respectively.

Conservation of Momentum:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial t} + v\frac{\partial u}{\partial t} + w\frac{\partial u}{\partial t} = g\beta(T - T_\infty) + \vartheta\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \qquad \cdots\cdots\cdots(1)$$

Conservation of Mass:

$$\frac{\partial \rho}{\partial t} + \rho\left(\frac{\partial u}{\partial x} + \frac{\partial u}{\partial y} + \frac{\partial u}{\partial z}\right) = 0 \qquad \cdots\cdots\cdots(2)$$

Conservation of Energy:

$$\frac{\partial \mathrm{T}}{\partial \mathrm{t}} + u\frac{\partial \mathrm{T}}{\partial \mathrm{x}} + v\frac{\partial \mathrm{T}}{\partial \mathrm{y}} + w\frac{\partial \mathrm{T}}{\partial \mathrm{z}} = \alpha\left(\frac{\partial^2 \mathrm{T}}{\partial \mathrm{x}^2} + \frac{\partial^2 \mathrm{T}}{\partial \mathrm{y}^2} + \frac{\partial \mathrm{T}}{\partial \mathrm{z}^2}\right) \qquad \cdots\cdots\cdots(3)$$

Governing equations are made non-dimensional using following length and velocity scales

$$L^* = L/H \qquad H^* = H/H \qquad B^* = B/H$$

$$Re^* = ul/\vartheta \qquad t^* = t\alpha/L \qquad u^* = uL/\alpha$$

Therefore, dimensionless form of governing equations can be stated as following:

Conservation of Momentum:

$$\frac{\partial u^*}{\partial t^*} + u^*\frac{\partial u^*}{\partial x^*} + v^*\frac{\partial u^*}{\partial y^*} + w^*\frac{\partial u^*}{\partial z^*} = g\beta(T - T_\infty) + \vartheta\left(\frac{\partial^2 u^*}{\partial x^{*2}} + \frac{\partial^2 u^*}{\partial y^{*2}} + \frac{\partial^2 u^*}{\partial z^{*2}}\right) \qquad \cdots\cdots\cdots(4)$$

Conservation of Mass:

$$\frac{\partial \rho}{\partial t^*} + \rho\left(\frac{\partial u^*}{\partial x^*} + \frac{\partial u^*}{\partial y^*} + \frac{\partial u^*}{\partial z^*}\right) = 0 \qquad \cdots\cdots\cdots(5)$$

Conservation of Energy:

$$\frac{\partial T^*}{\partial t^*} + u^* \frac{\partial T^*}{\partial x^*} + v^* \frac{\partial T^*}{\partial y^*} + w^* \frac{\partial T^*}{\partial z^*} = \alpha \left(\frac{\partial^2 T^*}{\partial x^{*2}} + \frac{\partial^2 T^*}{\partial y^{*2}} + \frac{\partial T^*}{\partial z^{*2}} \right) \qquad \cdots\cdots\cdots (6)$$

III. NUMERICAL MODELING

A. Geometry and boundary conditions:

The geometry chosen in the study was a classical room. The computational domain associated to the geometry was made of entire room with extruded inlet and outlet for the tranquilization of the flow as illustrated in figure 1. As stated previously, horizontal and inclined inlet had considered as a purpose of evaluation.

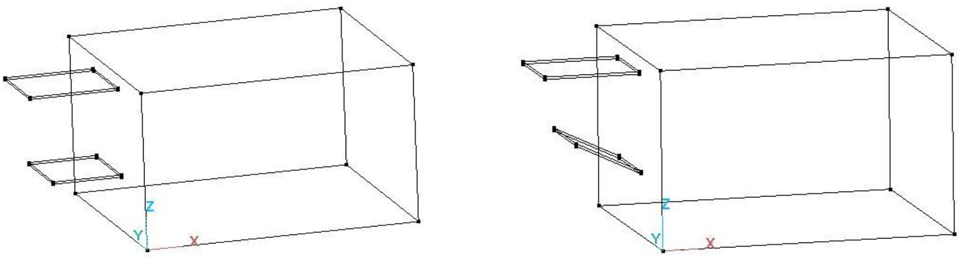

Figure 1(a): Classical domain (Left: Room with Horizontal Inlet; Right: Room with Inclined Inlet)

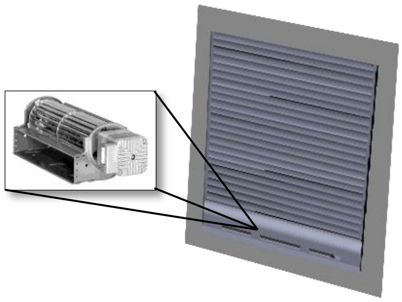

Figure 1(b): Window configuration

Inlet angle has been calculated by vector relation of velocity component for X and Z axis as following:

$$\tan\theta = \frac{v_z}{v_x} \qquad \ldots\ldots\ldots (7)$$

Adiabatic wall condition was applied to the numerical simulation of all (without wall thickness) domains. Domain with wall thickness was also considered to examine the effect of wall thickness on heat removal from the room. For simulation of domain with wall thickness, outside ambient temperature was applied to all the outer wall surfaces and the walls were considered as non-adiabatic except the floor of the room. A contact type interface had been used for proper heat transfer between solid walls and fluid region inside the room. Over and above, velocity inlet and pressure outlet was selected to the inlet and outlet of the domain respectively. Temperature distribution within the solid region of thick wall is governed by 1-D heat conduction equation that could be expressed as

$$\nabla^2 T = 0 \qquad \ldots\ldots\ldots (8)$$

At the interface between fluid region and solid region in the conjugate heat transfer model, the conductive heat transfer throughout the solid is coupled with the convective heat transfer in the fluid by

$$(\nabla\theta)_{fluid} = \frac{k_{wall}}{K} (\nabla\theta)_{wall}$$

Where θ is dimensionless temperature and k_{wall} is wall thermal conductivity

Moreover, for a large plane wall, one-dimensional (1-D) heat conduction equation can be applied and using the Fourier's law, the equation can be written as

$$Q_{cond} = -KA\frac{dt}{dx} \qquad \dots\dots\dots (9)$$

Where Q_{cond} is conductive heat transfer, K is thermal conductivity of solid material, $\frac{dT}{dx}$ *is* the temperature gradient, and A is the heat conduction area (m^2).

Thus, the total and surface heat fluxes can be evaluated by

$$q_{total} = \frac{T_{in,\infty} - T_{out,\infty}}{R_{total}} \qquad \dots\dots\dots (10)$$

$$q_{surface} = h_{in\ or\ out,\infty}\ (T_{in\ or\ out,\infty} - T_{inner\ or\ outer\ surface}) \qquad \dots\dots\dots (11)$$

Where, q_{total} is total heat flux (w/m^2), R_{total} is total thermal resistance (i.e., R-value) (m^2k/w), $q_{surface}$ is surface heat flux, h is heat transfer coefficient (w/m^2k) and indexes *in* and *out* are internal and external environments, subscript ∞ is ambient condition and *inner surface* and *outer surface* are internal and external surfaces of a finite-thickness wall.

Conceiving above theoretical aspect, we have considered horizontal inlet (Re= 1.7 x 10^3) domain with wall thickness. Concrete wall around room was considered 13 cm. For two different regions, two different models i.e. solid for walls and fluid for inner room space were chosen during numerical model section. Moreover, contact type interface was selected for conjugate heat transfer between fluid and solid model.

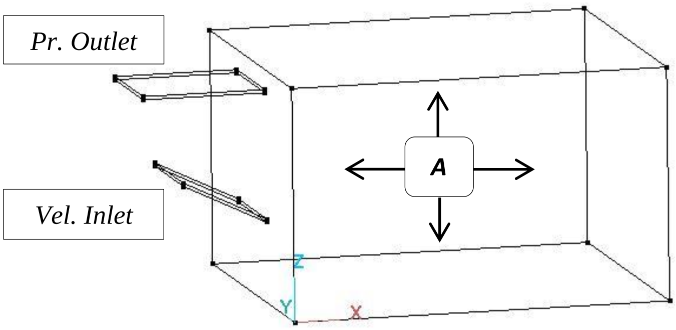

Figure 2: Domain's boundary condition without wall thickness; ***A*** = Adiabatic; Pr. = Pressure; Vel. = Velocity

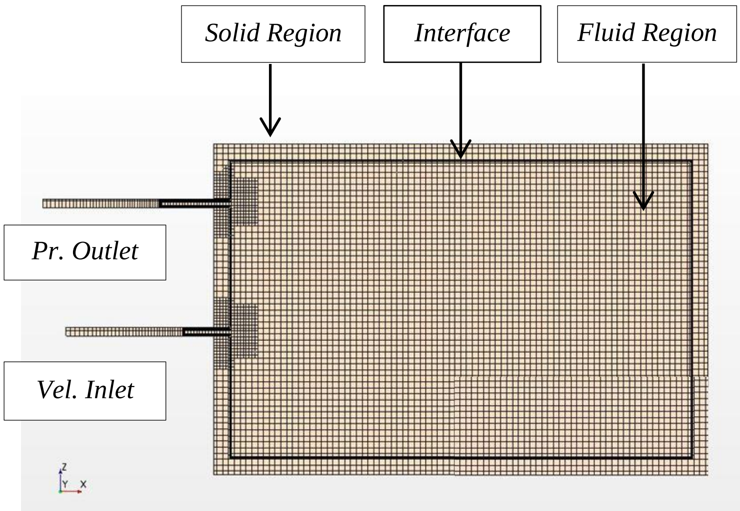

Figure 3: Domain's boundary condition with wall thickness; Pr. = Pressure; Vel. = Velocity

B. Grid independence analysis:

After developing 3D room geometry in the *StarCCM+* CFD software, trimmer hexa mesh was generated with prism layer and surface remesher. The optimal one was chosen by grid independent analysis. The discretization error was studied using the grid convergence index (GCI).

The GCI has proposed by *Roache* [8] for uniform reporting of grid convergence studies. The GCI is based on the *Richardson extrapolation* [9], where the discrete solutions are assumed to have a series representation of the discretization error. GCI converts the *Richardson extrapolation* error estimate into an uncertainty by using absolute values to provide an uncertainty bounds and using a factor of safety (Fs).

Solutions for fine, medium, and coarse grids are subscripted with n, n+1, and n+2, respectively. The GCI can be represented as:

$$GCI = \frac{F_s}{r^p - 1}\left|\frac{f_{n+1} - f_n}{f_n}\right| \qquad \dots\dots\dots (12)$$

where *r* is the grid refinement factor, *p* is the order of accuracy .

The grid refinement factor *r* for three-dimensional study is:

$$r = \left(\frac{N1}{N2}\right)^{1/3}$$ Where N is the number of cells or elements

The order of accuracy is then estimated by using the following equation:

$$p = \frac{ln\left(\frac{f_{n+2} - f_{n+1}}{f_{n+1} - f_{n+1}}\right)}{\ln(r)}$$

Several domains with different grid number such as $100 \text{ x } 10^3$, $200 \text{ x } 10^3$, $400 \text{ x } 10^3$, $600 \text{ x } 10^3$ was tested to find the optimal grid for numerical solution.

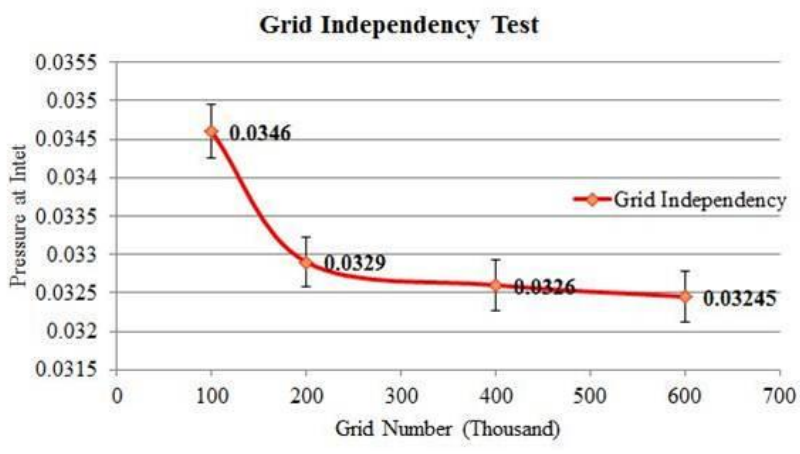

Figure 4: Grid independence test by Pressure (*Pa*) at the inlet. The error bar in the plot shows 1% error.

As the exact error for grid $400 \text{ x } 10^3$ and $600 \text{ x } 10^3$ was 0.9 % and 0.6%, while the GCI was 0.33% and 0.25% respectively. We have chosen the $400 \text{ x } 10^3$ grid to reduce computational cost especially in the unsteady simulation.

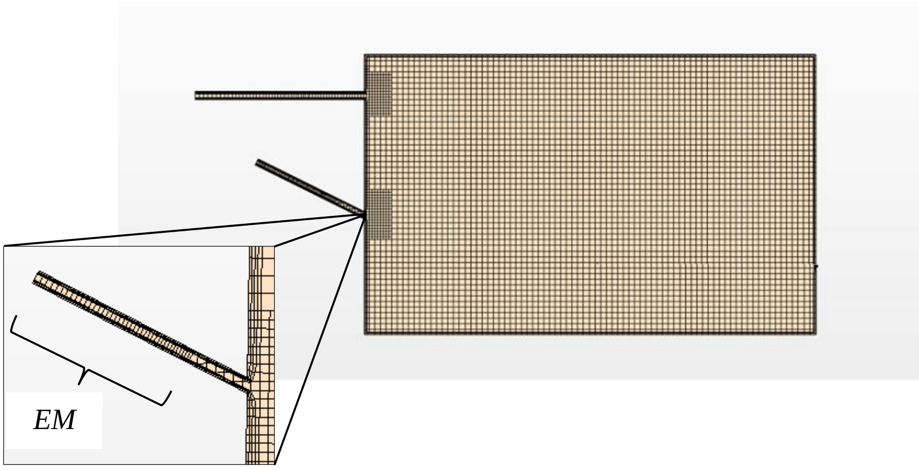

Figure 5: Mesh (400 x 10^3) of room with inclined inlet (EM: Extruder Mesh)

Moreover, Extruder mesh has been built in the inlet so that the velocity profile can develop up to the desired value at the domain inlet surface. Same procedure was followed to outlet for pressure according to StarCCM+ user guide.

C. *Numerical model selection:*

For solving the convection problem unsteady, incompressible, turbulent (*k-ε)* model, due to Rayleigh number 7.3 x 10^{11}, was chosen. Moreover, RANS and realizable two layer additional model had also considered according to literature review and Star CCM+ user guide. Firstly, Ideal gas with segregated flow, due to incompressible fluid, was selected according to user guide and then the same problem with coupled flow had also solved to identify accuracy of numerical solution. Moreover, Computation had been made with finite volume code. Second order upwind scheme was used in the discretization of governing equation. SIMPLE algorithm is used to link the pressure and velocity field. First order time implicit scheme has been used to resolve the unsteady term. The time step used in the computations is 1.0 s.

D. *Time step independence analysis:*

An optimal time step was chosen by comparing the three different, such as 0.5, 1.0, 2.0, solver time steps. For each time step the result was same except the simulation time needed to compute 30 minutes operation of system by *Intel Xenon processor*. Finally we have decided to consider the solver time step 1.0 to save time.
Time step has been calculated by following formula

$$CFL\ number, C = \frac{u\Delta t}{\Delta x} \leq C_{Max} \qquad \text{......... (13)}$$

Where, Δx is the spacing of the grid; Δt is the time step and *u* is the velocity.

The value of C_{max} is dependent on the method used to solve the system of algebraic equation (Explicit/Implicit). For explicit solver, C_{max} = 1; for Implicit on the other hand, C_{max} may be a little bigger number, but it wouldn't go beyond 2.

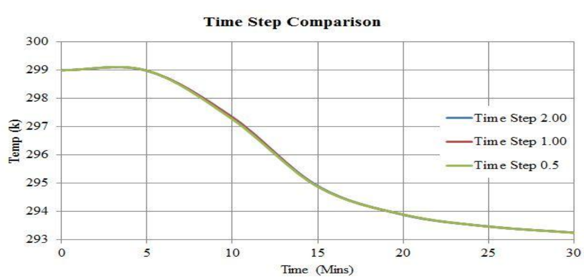

Figure 6: Time step comparison by Temperature (*k*) at 1.5m from the floor

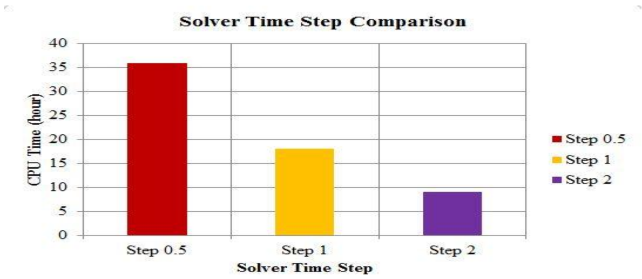

Figure 7: CPU time comparison

IV. RESULTS ANALYSIS

Numerical simulations were performed with domain (room) to verify the effect of inlet angle. The comparison between two different configurations was made by heat transfer and bulk temperature considered on different horizontal plane between floor and ceiling. By definition Bulk temperature is the mass weighted average temperature at a given axial location/plane.

Bulk temperature can be defined as:

$$T_b = \frac{1}{AU_m} \int_A uT \ dA \qquad \dots \dots \dots (14)$$

Where the mean velocity can be expressed as:

$$U_m = \frac{1}{A} \int_A u \, dA$$

We have chosen bulk temperature as a function of 'z' axis only in order to evaluate the mixing and therefore the average thermal comfort along the height of the room to evaluate the effect of geometrical and dynamic parameters on the characteristic time and uniformity of heat removal in the room.

A. Flow structure with temperature distribution:

Streamlines has been determined inside domain to realize flow structure and temperature distribution. Initially lower Reynolds number (*$Re=8.6 \times 10^2$*) represents heat removal just at the lower portion of domain after successful termination of air refreshment system (30 minutes). Though the lower portion of the room temperature was decreased to outside ambient temperature, remaining part of the domain possess maximum temperature. Hence, we have increased velocity of the inlet by remaining other parameters as earlier that theoretically also increased the Reynolds number (*$Re=1.7 \times 10^3$*). Finally, the new results were one step more close to the desired goal than earlier. So, It conveyed exceeding heat removal.

<u>Case 1</u>: <u>*$Re=8.6 \times 10^2$*:</u>

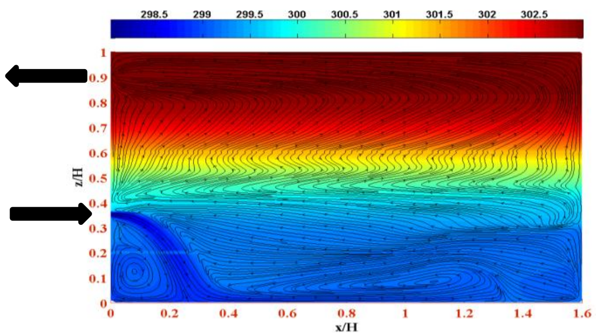

Figure 8: Horizontal inlet

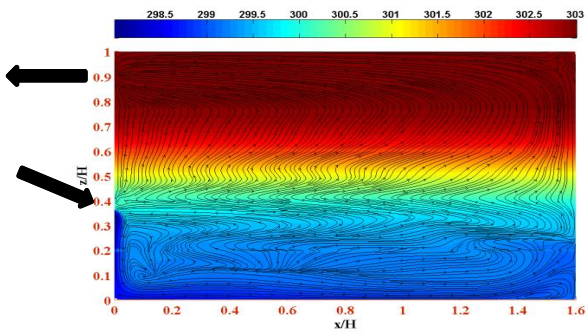

Figure 9: Inclined inlet

<u>Case 2</u>: <u>*$Re=1.7 \times 10^3$*:</u>

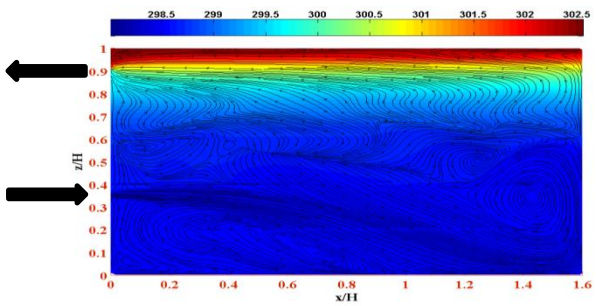

Figure 10: Horizontal inlet

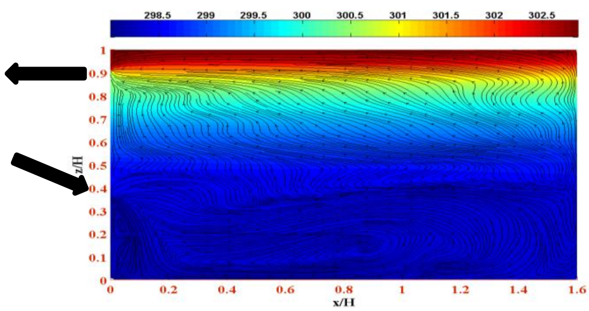

Figure 11: Inclined inlet

B. *Inlet angle comparison:*

Horizontal and Inclined inlet has compared by bulk temperature at different horizontal planes to find out the optimal selection of inlet for maximum heat removal from domain. For lower Reynolds number ($Re = 8.6 \times 10^2$), inclined inlet (dotted Line) represents lower temperature than horizontal inlet (solid line) up to 1.5 m from the floor. However, temperature pattern has inverted from above 1.5 m to rest of the domain. Similar tendency has been observed in Reynolds number, $Re = 1.7 \times 10^3$. Moreover, increment of Reynolds number helped to remove additional heat from domain that was also prominent in flow structure and temperature distribution graph.

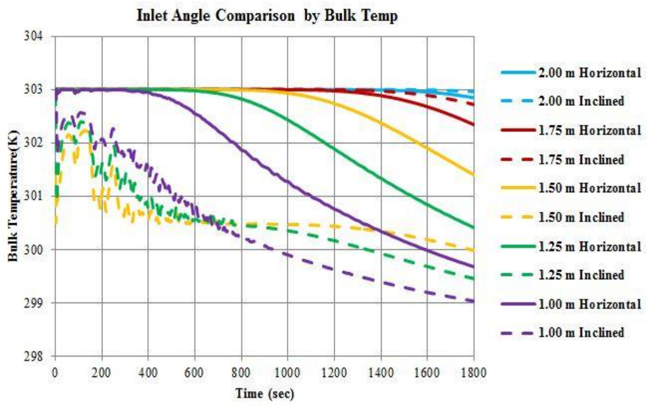

Figure 12: Horizontal inlet, $Re = 8.6 \times 10^2$

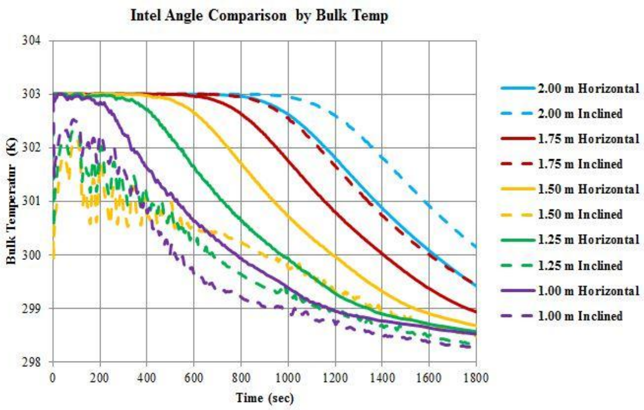

Figure 13: Inclined inlet, $Re = 1.7 \times 10^3$

Therefore, from the comparison of inlet angle effect it can be conclude that bulk temperature of different planes indicates, inclined inlet could be suitable from floor to 1.50 m for lower temperature while horizontal inlet for rest of the part (above 1.50 m).

C. *Total heat transfer:*

Heat transferred or removed from the domain was also considered to identify more reliable results because the more heat is transferred or removed from the domains then the thermal comfort will increase. We have calculated all the domains for segregated flow solver with ideal gas and heat transfer has been calculated throughout the simulation duration (1800 seconds). Then summation of heat transfer at every second mathematically leads to total heat transfer from domains. Furthermore, those two parameters have been compared not only graphically but also numerically to verify the accuracy by absolute and relative error.

Heat flux removed from the insulated room between the inlet and the outlet by the air flow can be defined as follow:

$$\dot{Q} = \dot{m} * C_p * \Delta T \qquad \dots\dots\dots (15)$$

Then total heat removed from the room during an operation period of time is then

$$Q_{total} = \int_{t=1}^{t=1800} \dot{Q}\, dt \qquad \dots\dots\dots (16)$$

Graphical representation of heat transferred from domain throughout the air refreshment process has illustrated below

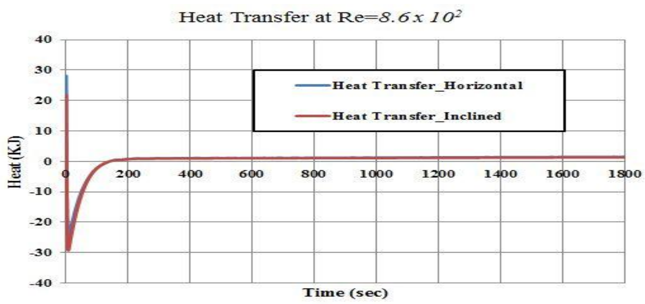

Figure 14: Heat transfer (*KJ*), *Re*=8.6 x 10^2

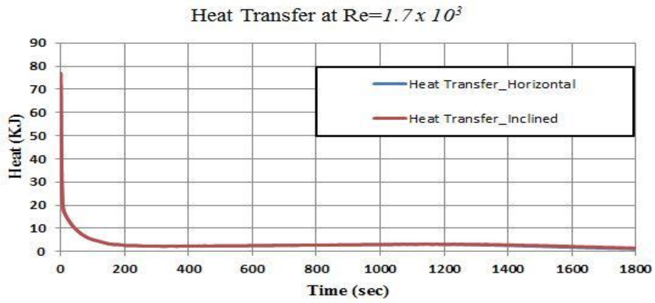

Figure 15: Heat transfer (*KJ*), *Re*=1.7 x 10^3

Table 1: Inlet angle effect comparison by total heat transfer

Inlet types / Reynolds number	Horizontal inlet (KJ)	Inclined inlet (KJ)	Absolute difference (KJ)
Re = 8.6 x 10^2	948.092	517.852	466.24
Re = 1.7 x 10^3	5556.918	5772.698	215.78

Hence, above comparison of total heat transfer shows that horizontal inlet removes comparatively more heat than inclined inlet.

D. Wall thickness effect:

The effect of wall around domain has been observed by drawing disparity of bulk temperature and total heat removal with a domain without wall thickness. As we have seen from the previous result analysis that horizontal inlet with *Re=1.7 x10^3* was close enough to the desired goal of air refreshment system, we have considered those parameters to analyze wall thickness effect. Now if we focus on quantitative results by comparing bulk temperature at different horizontal planes for both configurations, we have seen that there was a significant temperature variation after 30 minute of heat removal. Furthermore, the temperature decreases negligibly even after one hour of heat removal for wall thickness effect.

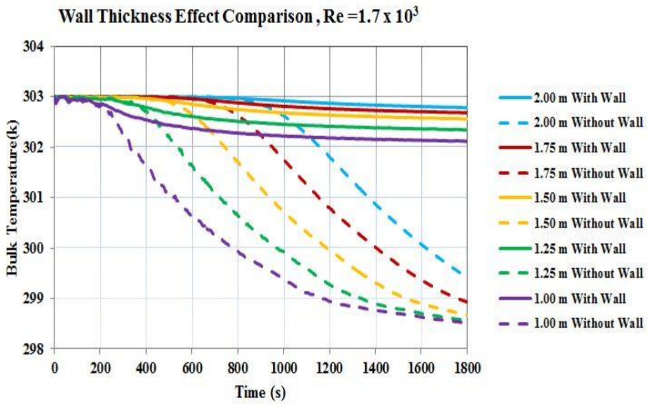

Figure 16: Wall thickness effect comparison by bulk temp

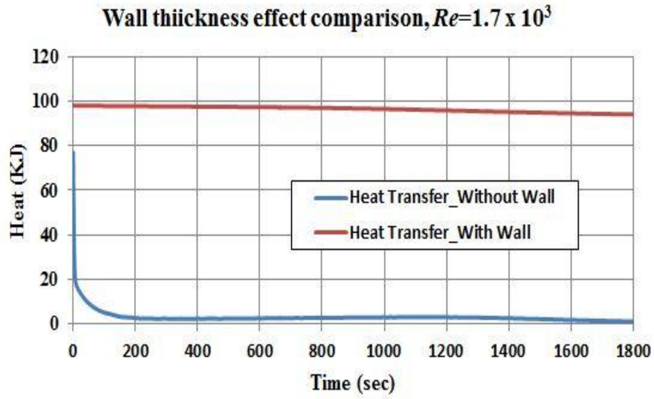

Figure 17: Wall thickness effect comparison by heat transfer

Therefore, we can easily conclude that we need more time for desired heat removal with wall thickness. The time for heat removal would be a function of wall materials, thickness, insulation type and some other parameters corresponding to different phenomenon.

V. CONCLUSION

A classical empty room was assumed as domain with horizontal and inclined inlet for summer season and the temperature range had been considered for southern region of France. Then the effect of inlet angle for air refreshment has been observed by numerical simulation to acquire optimal solution for the system. Meanwhile, Mixed mode (Mechanical ventilation through inlet, Natural ventilation in room) ventilation method had considered as principle of heat removal from room. As the Natural ventilation is a major part of Mixed-mode ventilation and optimally operate in small temperature difference, the system will function 30 minutes just in night when the outdoor temperature is lower than indoor A mechanical fan was placed for forced ventilation at the inlet which located below the window and the outlet is placed just upper portion of window. We have simulated domains with maximum critical conditions so that we can ensure about the system's highest ability regarding heat removal. Therefore, window remained closed because sometime it's might not possible to open window due to dust, noise or insects especially room located in ground floor. Moreover, the door was also considered closed and there was no air pass except outlet. Numerical analysis has been performed with Reynolds Number 8.6×10^2, 1.7×10^3 by considering effect of inlet angle (horizontal, inclined) and wall thickness. Finally, all the parameters have compared by bulk temperature on horizontal planes and heat transferred from domains to find the optimum solution.

For inlet angle effect, Inclined inlet represents lower temperature than horizontal inlet up to 1.5 m from the floor in lower Reynolds number. However, temperature pattern has inverted above 1.5 m from the floor up to rest of the domain. Similar tendency has been observed in Reynolds number 1.7×10^3. Moreover, increment of Reynolds number helped to remove additional heat from domain that was also prominent in flow structure and temperature distribution graph. From the comparison for inlet angle effect it can be conclude that bulk temperature of different planes indicates, inclined inlet could be suitable from floor to 1.50 m for lower temperature while horizontal inlet for rest of the part (above 1.50 m).

In case of wall thickness consideration, we can easily conclude that we need more time for desired heat removal with wall thickness. The time for heat removal would be a function of wall materials, thickness, insulation type and some other parameters corresponding to different phenomenon.

VI. ACKNOWLEDGEMENT

First of all, I would like to be grateful to the Almighty for bestowing me the strength and patience to complete the project successfully. I would also like to thank *Mr. Yannick HOARAU* (Professor, Computational Engineering Dept., *Université de Strasbourg*, France) for his encouragement, guidance and beneficial criticize on this project. Finally, I solemnly acknowledge that this project wouldn't be successful without financial and technical support provided by Energy, Electricity and Automation Department, *Hautes Etudes d'Ingénieur* –Lille.

VII. REFERENCES

[1] Y. Jiang, D. Alexander, H. Jenkins, R. Arthur, Q. Chen, Natural ventilation in buildings: measurement in a wind tunnel and numerical simulation with large eddy simulation, Journal of Wind Engineering and Industrial Aerodynamics 91 (3) (2003) 331–353.

[2] G. Evola, V. Popov, Computational analysis of wind driven natural ventilation in buildings, Energy and Buildings 38 (5) (2006) 491–501.

[3] Jin M, Zuo W, Chen Q. Simulating natural ventilation in and around buildings by fast fluid dynamics. Numerical Heat transfer, Part A: Applications, An International Journal of Computation and Methodology 2013; 64:273-289.

[4] T. Catalina, J. Virgone, F. Kuznik. Evaluation of thermal comfort using combined CFD and experimentation study in a test room equiped with a cooling ceiling. Building and Environment, Elsevier, 2009, 44 (8), pp.1740-1750.

[5] G.M. Stavrakakis, Vrachopoulos Michail, N.C. Markatos, Natural Cross-Ventilation in Buildings: Building Scale Experiments , Numerical Simulations and Thermal Comfort Evaluation; Energy and Buildings · January 2008

[6] Kuznetsov GV, Sheremet MA. Conjugate natural convection in an enclosure with a heat source of constant heat transfer rate. International Journal of Heat and Mass Transfer 2011; 54:260-268.

[7] Horikiri, K., Yao, Y. and Yao, J. (2014) Modeling conjugate flow and heat transfer in a ventilated room for indoor thermal comfort assessment. Building and Environment, 77. pp. 135-147. ISSN 0360- 1323

[8] Roache P. J.(1994). Perspective: A method for uniform reporting of grid refinement studies. International Journal of Heat and Fluid Flow 24 54-66

[9] Richardson L., Gaunt A. (1927). The Deferred Approach to the Limit. Part I. Single Lattic. Part ii. Interpenetrationg Lattices. The Philosophical Transcations of the Royal Society of London Series A, Containing papers of a Mathematical or Physical Character, 116 (3):405-41

[10] A. J. Chorin (1968). Numerical solution of navier-stokes equations. Mathematics of Computation 22 745-762

[11] ASHRAE(2010). ASHRAE Standard 55-2010. Thermal Environment Standards for Human Occupancy Atlanta, GA

[12] V. Antony Aroul Raj , R. Velraj; Review on free cooling of buildings using phase change materials; Renewable and Sustainable Energy Reviews 14 (2010) 2819–2829

[13] Geros V, Santamouris M, Karatasou S, Tsangrassoulis A, Papanikolaou N. On the cooling potential of night ventilation techniques in the urban environment. Energy and Buildings 2005;37:243–57.

[14] Convention problem modeling, STAR CCM+ user guide.