2018 Building Performance Analysis Conference and
SimBuild co-organized by ASHRAE and IBPSA-USA
Chicago, IL
September 26-28, 2018

PREDICTION OF IMPROVED OCCUPANT'S THERMAL COMFORT WITH CEILING FAN THROUGH COUPLED ENERGY SIMULATION AND COMPUTATIONAL FLUID DYNAMICS

Brijesh Pandey[1], Atul Sharma[2] and Rangan Banerjee[1]
[1]Department of Energy Science and Engineering, IIT Bombay
[2]Department of Mechanical Engineering, IIT Bombay
Email: brijeshpandeyby715@gmail.com

ABSTRACT

Ceiling fans are considered as effective energy saving option for occupant's thermal comfort improvement. In this study, a generic method is developed to integrate the effect of air flow induced by a ceiling fan in a whole building simulation model to quantify its effect on occupant's thermal comfort and associated reduced cooling energy demand. A co-simulation between EnergyPlus and Computational Fluid Dynamics (CFD) has been performed. A generalized interface is developed using Matlab and Building Control Virtual Test Bed (BCVTB) to couple EnergyPlus and Fluent for co-simulation. A cubical room comprising a window having dimensions of 3m×3m×3m and 2m×2m respectively and equipped with a ceiling fan is taken to illustrate the applicability of the method. Simulation is performed for the typical summer day of Mumbai city of India. This paper concludes that for considered case using ceiling fan improves thermal comfort index by 38% and reduces cooling load by 27%.

INTRODUCTION

Achieving thermal comfort, while minimizing the energy use is the goal of building research. Thermal comfort of the occupant depends on temperature, air movement, relative humidity, occupant's metabolic rate and occupant's clothing insulation value. From the above-mentioned factors, air movement surrounding the occupant has been shown as a method of thermal comfort improvement in warm environments. A recent reanalysis of ASHRAE database of the air-conditioned building shows that occupants want higher air movement when their thermal sensation get slightly warm or warmer. Recently ASHRAE standard 55-2010 "Thermal environment conditions for human occupancy" increased the allowable air movement inside the building for thermal comfort in warm environments. Natural ventilation is an energy efficient option of increasing air movement in an occupied zone but it does not always guarantee the specified air movement for comfort. Ceiling fans were shown to be an energy efficient method of providing air movement for improving occupants' thermal comfort in literature.

Early laboratory studies (e.g. Rohles et al. 1983 and Scheatzle et al. 1989) show that air flow induced by a ceiling fan with a velocity between 0.5 and 1 m/s compensated for 2.8-3.3 °C temperature change and resulted in energy saving of 15-18%. A relationship between building cooling energy and thermostat setpoint temperature with ceiling fans is discussed in Mortan-Gibson et al. 1985 and James et al. 1996. It is concluded that increased set point by 2 °C with ceiling fan saves cooling energy without compromising the thermal comfort of the occupants.

Son et al. 2005 studied numerically, the effect of a ceiling fan on thermal comfort using predicted mean vote (PMV) as thermal comfort index for 2D and 3D room. It is reported that ceiling fan induced air, forced the thermal comfort index towards the cooler region. The whole building simulation model is not considered in this study to get factors e.g. occupancy schedule, metabolic rate, relative humidity, clothing value and mean radiant temperature to calculate PMV which limits this study.

Recent laboratory studies (e.g. Zhang et al. 2010, McIntyre et al. 2012, Zhai et al. 2015) also show the potential of the ceiling fan in improving the thermal comfort of indoor environment for warm and warm and humid climates. It is concluded in these studies that air flow induced by ceiling fan improves thermal sensation and perceived air quality at an elevated temperature of up to 30 °C and 50% RH. In spite of having much potential for thermal comfort improvement, the effect of ceiling fan has not been incorporated in whole building simulation tools. There is no generic method available in the literature to predict the air flow induced by ceiling fan on thermal comfort and associated reduced cooling energy demand through building modeling simulation tools.

In this paper, a method is proposed to integrate the effect of air flow induced by a ceiling fan in a whole building simulation model to predict its effect on thermal comfort and cooling energy for early building design stage. An automated iterative co-simulation between nodal model based building simulation tool, i.e., EnergyPlus (version 8.5, Crawley et al. 2001) and Finite volume method (FVM) based tool, i.e., Fluent (version 14.5, Fluent 2014) is proposed to capture the effect of ceiling fan on thermal comfort and the associated cooling energy reduction.

Co-simulation method is proposed because nodal model based EnergyPlus can predict the building energy consumption but has the limitation of not being able to simulate the effect of air recirculation induced by ceiling fan while CFD has the capability to model this phenomenon. However, limitations also come with the CFD, whole building system modeling in CFD requires a lot of computational time and also there will be need of boundary conditions to initiate the simulation. While working with solid-fluid model combined, i.e. conjugate heat transfer problem, solutions become unstable most of the time due to the different response time of the solid and fluid material. Generally, the response time of the building's surfaces is much higher than the adjacent fluid which makes the system stiff. To predict the behavior of whole building system quickly by incorporating the effect of fluid flow a co-simulation method is considered in this paper.

A coupling method is required to perform the co-simulation between the nodal based approach and CFD. Nagai et al. 2009 did the manual coupling of the CFD and nodal model to perform the co-simulation but the disadvantage of this method is that the effect of CFD can't be incorporated in the nodal model or vice-versa at each time step. This problem can be avoided through automated iterative internal or external coupling methods. Negrao et al. 1998 used ESP-r and CFD for automated iterative co-simulation of the nodal method and CFD using internal coupling approach. Capability of ESP-r has been extended by implying a conflation controller to configure CFD model at each time step. Disadvantage of this method is that both nodal model and CFD can't be tuned individually whatever changes applied to the one would affect the other. Internal coupling method has slow convergence and it is also very programming intensive.

Djunaedy et al. 2004, Chen et al. 1995 and Zhai et al. 2001 have worked on the advantages and disadvantages of internal coupling and concluded that external coupling of CFD and nodal method was more favorable because it avoided the stiffness issue and was computationally less expansive. Wang et al. 2008 and Zhang et al. 2013 performed external coupling between nodal model and CFD, variables were exchanged through interface between the two solvers and results were compared with uncoupled simulation and it was found that coupled strategy gave better results.

In this study, a new approach and interface is developed to couple EnergyPlus and Fluent using Matlab R2014b and BCVTB (Wetter et al. 2011) for the iterative automated co-simulation.

A cubical room of dimensions 3m×3m×3m equipped with hydronic radiant heating ventilation and air conditioning (HVAC) system and a ceiling fan has been taken for the co-simulation demonstration. Inside surfaces temperature, mean air temperature, mean radiant temperature, clothing value, relative humidity, surface heat transfer coefficient and PMV value are selected for the exchange variables.

PMV value is calculated with above-stated variables exchanged by EnergyPlus and air velocity transferred by Fluent at each time step of the co-simulation. PMV value is sent back to EnergyPlus to actuate the room setpoint temperature at each time step.

Parameters like mean air temperature, PMV and cooling energy consumption are compared for the co-simulation and base case, i.e., without ceiling fan computed with EnergyPlus.

COUPLING METHOD

Coupling strategy between nodal model and CFD can be categorized into three methods which are internal coupling, external coupling and external coupling with sequential simulators execution. Negrao et al. 1998 shows that internal and external coupling strategies require a large number of iterations to reach convergence.

In this study, external coupling with sequential simulators execution has been used to perform the co-simulation because of its capability to individually tune the time step of the nodal model and CFD and faster convergence. EnergyPlus (version 8.5) and Ansys Fluent (version 14.5) have been used for building simulation and CFD simulation respectively. An interface has been developed using BCVTB and Matlab R2014b to couple Fluent and EnergyPlus. An addition Matlab script file is written to exchange variables at each time step of co-simulation. Figure 1 explains the applied external coupling; it can be inferred that EnergyPlus will start the simulation at time step t_n and halts till it receives data back from interface. The exchange variables will be transferred to the coupling interface though the external interface of EnergyPlus via BCVTB using configuration file, then a Matlab script will extract the data from the interface and supply it to the respective boundary of CFD to start the simulation at same time step t_n. After CFD solution convergence, exchange variables will be extracted and sent to the interface that will be further communicated to EnergyPlus through BCVTB to start the simulation for the next time step $t_{n+\Delta t}$.

EXCHANGE VARIABLES

Inside wall surface temperature from EnergyPlus is transferred through the external interface to the CFD for the wall boundary condition and area-weighted average surface heat transfer coefficient is sent back to EnergyPlus. Additional variables like mean radiant temperature, relative humidity, occupant's clothing value and metabolic rate are exchanged through external interface of EnergyPlus and at the same time mean air temperature and air velocity are extracted from the CFD simulation to calculate the thermal comfort index, i.e., PMV and sent back to EnergyPlus to actuate the room setpoint temperature at each time step.

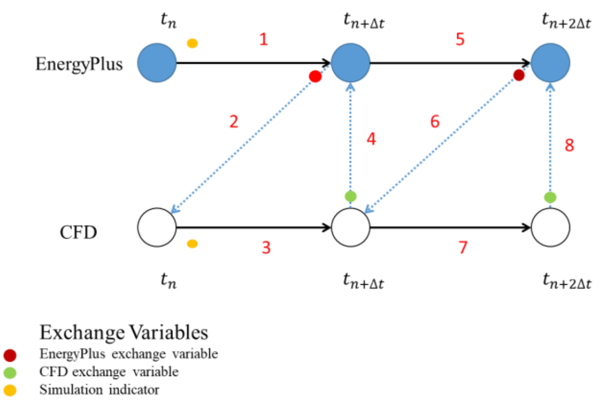

Figure 1 External coupling with sequential simulation execution strategy of nodal method of EnergyPlus and CFD (Fluent)

COUPLING PLATFORM

A coupling platform is developed using Matlab R2014b and BCVTB for the co-simulation between EnergyPlus and Ansys Fluent. BCVTB is a software environment targeted to provide integration between various simulation tools like EnergyPlus and Matlab/Simulink, EnergyPlus and Trnsys and EnergyPlus and Dymola. For example, in this study BCVTB enables concurrent EnergyPlus simulation for the whole building with HVAC system and CFD simulation in Fluent, while exchanging data between the two at each time step.

ENERGYPLUS OBJECT IN SUPPORT OF CO-SIMULATION

Two EnergyPlus input objects called "ExternalInterface: Actuator" and "ExternalInterface: Variable" are used for data exchange. "ExternalInterface: Actuator" is used to apply the area weighted average surface heat transfer coefficient of walls obtained from the CFD simulation tool and "ExternalInterface: Variable" is used to get the PMV value calculated externally with various exchanged variables of EnergyPlus and CFD through another program. Another EnergyPlus object called Energy Management System (EMS) tool is used to actuate the set point temperature of the occupied zone sensing the variable, i.e., PMV value obtained from "ExternalInterface: Variable."

PROGRAM FOR EXECUTING CFD SIMULATION AND EXTRACTING EXCHANGE VARIABLES

A Matlab script is written to collect EnergyPlus variables at each time step transferred through the external interface of EnergyPlus and write in a text file, execute the Fluent software to perform CFD simulation and after convergence of CFD simulation extract needed variables and send back to EnergyPlus through BCVTB. A user-defined function is written in C for reading the text file written by the previous program and applying the appropriate boundary condition for CFD simulation. A journal file of Fluent is written to read the case file of CFD and user-defined function is hooked up in this to automatically read the text file of exchange variables and apply boundary condition for CFD at each time step of co-simulation. Another Matlab script is written to calculate the thermal comfort index, i.e., PMV value after finishing off the CFD simulation from variables obtained from EnergyPlus and Fluent. The flow chart of the above-described procedure is shown in Figure 2.

CONFIGURATION FILE FOR EXCHANGE VARIABLES

The configuration file is an important component in co-simulation to exchange the variables from EnergyPlus to BCVTB and in reverse order. Configuration file follows the XML file format. Variables written in the configuration file are considered as a sequence of the array and transferred in the same order in which state variables are written. A Matlab script will then collect the variables from BCVTB and transfer it to the CFD. After the convergence of CFD solution script will collect data and transfer back to the BCVTB which will be further passed to the EnergyPlus in the same order.

CO-SIMULATION PROCESS

Co-simulation process is shown in Figure 2. The newly developed interface of Matlab using BCVTB takes EnergyPlus input data file, weather file and journal file of Fluent as input and invoke the co-simulation by calling EnergyPlus and Fluent. Firstly, EnergyPlus is executed for the whole building simulation for the first time step, after EnergyPlus finishes the simulation for the first time step, data is sent through the BCVTB to the Matlab script and EnergyPlus halts till it receives exchange variables back from BCVTB for next time step. After receiving the data from EnergyPlus Fluent is executed to conduct steady-state CFD simulation till its convergence. The exchange variables, i.e., heat transfer coefficient and PMV value are calculated and sent back to EnergyPlus through BCVTB. The above process makes one complete cycle

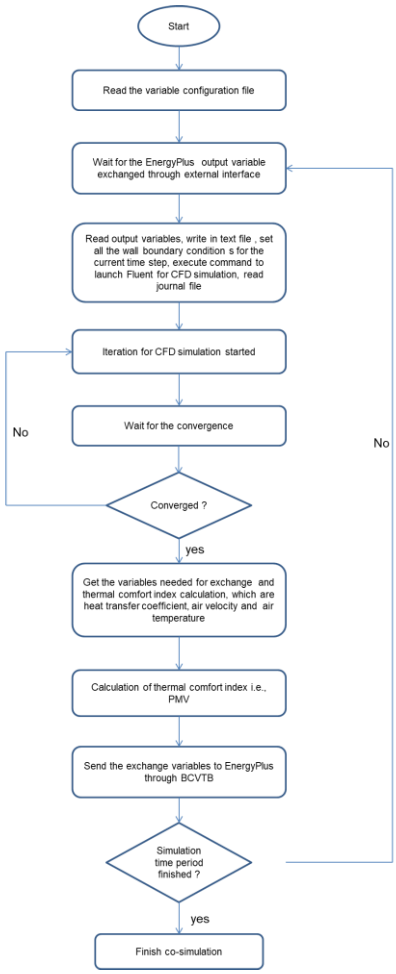

Figure 2 Flow chart of co-simulation process

After receiving back, the exchange variables, EnergyPlus starts simulation for the next time step for the next cycle and process goes on until the EnergyPlus simulation time period finished.

CASE FOR CO-SIMULATION

The proposed method and developed interface is very generic and will work for any types or size of the building. To demonstrate the applicability of method and interface a single-storey building of dimensions 3m×3m×3m shown in Figure 3 is considered for the co-simulation. EnergyPlus input data file for this building has been created using EnergyPlus 8.5. Weather file is incorporated in epw format for the Mumbai, city of India. To create the EnergyPlus input data file, building is considered as fully air-conditioned. Radiant hydronic system is taken as HVAC system. Time step for EnergyPlus simulation was set as 60 min. Table 1 and 2 provides the thermal and electrical loads and construction details of the building respectively. Ceiling fan is considered in the CFD model of the building. CFD model is created using Ansys Fluent and shown in Figure 6. First, summer design week is analyzed and 21st May is considered to perform the co-simulation because of the worst outdoor condition. PMV value is taken as thermal comfort index to see the effect of ceiling fan on occupant's thermal comfort. PMV value is calculated at the height of 1m from the floor taking the variables, air velocity induced by ceiling fan and air temperature at plane 1m above the floor from CFD simulation and mean radiant temperature, relative humidity, clothing value and metabolic rate from EnergyPlus. This PMV value is used to actuate the set point temperature of the building to see the effect of the ceiling fan on HVAC cooling energy consumption.

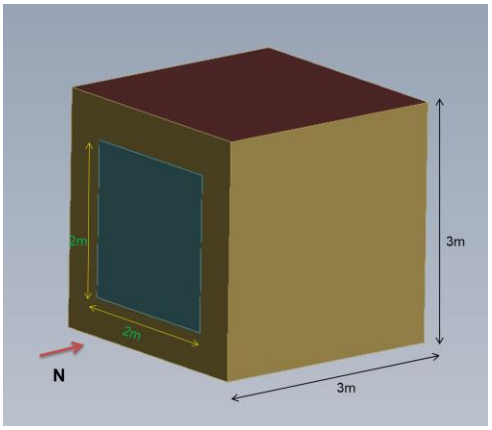

Figure 3 Computational domain for EnergyPlus model

Table 1 Thermal and electrical load of the selected building

Load/System	Rating and Description
Occupancy	3 Occupants with a specified schedule
Electrical load	2928 W
HVAC System	Radiant conditioning system with no air loop, thermostat setting at 20 °C in winter with 15 °C setback temperature, 24 °C in summer. No outside air

Table 2 Construction details of the building

Surface Name	Construction Details
Walls	2 layered wall, layer 1 is 0.1m common brick, layer 2 is 0.019m plaster
Roof	3 layered construction, 1^{st} layer is 0.009m membrane, 2^{nd} layer is 0.025m insulation and 3^{rd} layer is 0.05m heavy weight concrete
Floor	2layered construction, 1^{st} layer is 0.1m concrete and 2^{nd} layer is 0.001m finish flooring-tile
Window	The window is 3mm clear glass. Window to wall ratio is 70%.

CFD model

Input details of the CFD model is given in table 3. The CFD model of the selected building has the same geometric setup as in EnergyPlus model. A ceiling fan is incorporated in the CFD model. Air flow induced by ceiling fan has been modeled with two computational domains one is rotating domain and other one is stationary domain. The stationary domain has dimensions of the selected building, i.e., 3m×3m×3m. The dimension of the rotating domain has been decided with a number of trial simulations. The height and radius of the rotating domain have been selected as 0.22m and 0.66m respectively. The ceiling fan under consideration is of diameter 1.3m with a hub height and diameter of 0.047m and 0.24m respectively and rake angle 8° shown in Figure 4.

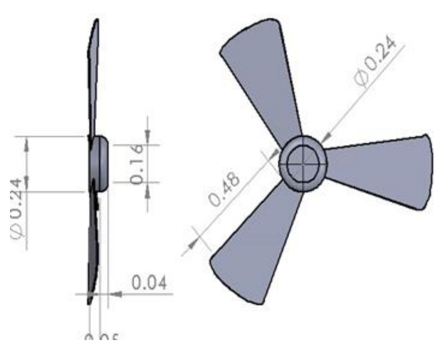

Figure 4 Solid model of the ceiling fan

Rotating domain is created in the middle of the room and 2.5m above the floor. Unstructured tetrahedral mesh element has been used for both the stationary domain and rotating domain. Since there is an interface between the stationary and rotating domain, sharp velocity change occurs at the interface. To capture the real air flow induced by ceiling fan, body sizing element has been used for the rotating domain for further refinement of mesh element. A grid independent study has been performed to decide the mesh sizing of the computational domain. Velocity at the interface of rotating and stationary domain is taken as a parameter for grid independence because of a sharp change in its value at the interface. Figure 5 shows that after 4.6 million mesh element there is little change in velocity so a total number of mesh elements considered are 4.6 million for this study. Approximately 97% of the overall mesh elements are located in the rotating domain.

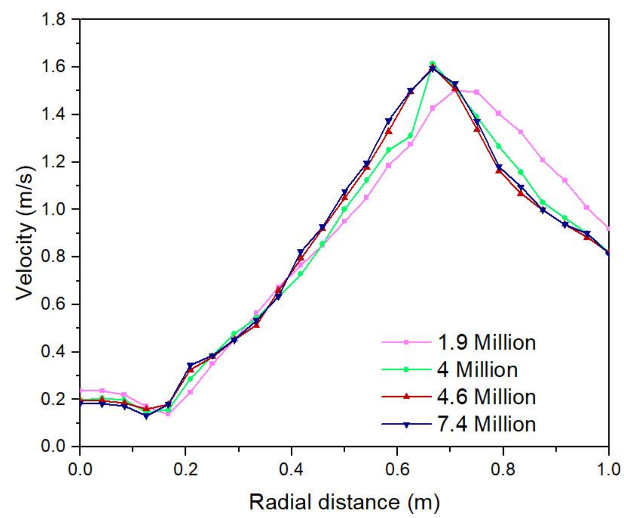

Figure 5 Grid independent study

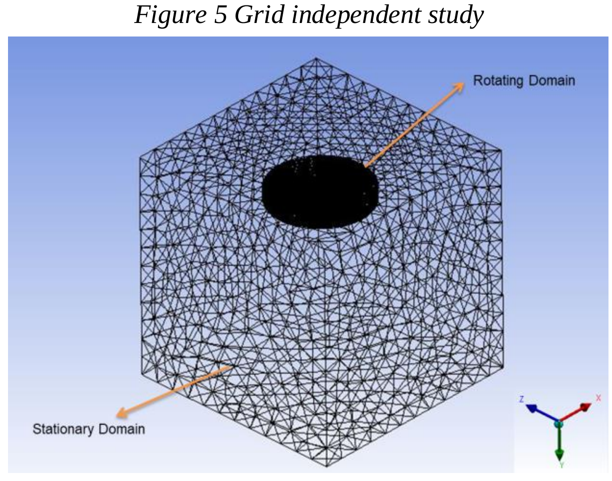

Figure 6 Computation domains of CFD model and grid structure

Table 3 Inputs for the CFD simulation

Input variables	Description
Stationary domain dimensions	3m×3m×3m
Rotating domain dimensions	1.32m diameter and 0.22m height
Mesh structure	Unstructured Tetrahedral and body sizing of mesh for the rotating domain
Rotating speed	160 rpm
Wall boundary condition	No slip boundary condition on all the six faces of the stationary domain and surfaces of the ceiling fan
Model for Rotating domain	Rotating frame reference model
Turbulence model	Shear Stress Transport (SST) model
Simulation type	Steady State
Working fluid	Air at 25 °C
Advection Scheme	First Order Upwind
Convergence criteria	0.001

Figure 6 shows the computational domain and grid structure of the CFD model. Moving reference frame method has been selected to model the rotating domain. In this method, solid model of the ceiling fan in the rotating domain does not rotate, but the mesh around the ceiling fan moves with the given rotating speed with respect to the stationary ceiling fan. This method has been selected for the faster convergence. No slip boundary condition has been applied to all the wall of stationary domain and surface of the ceiling fan. Shear Stress Transport (SST) turbulence model has been selected to simulate the flow induced by the ceiling fan because of its faster convergence for turbomachinery problem. Upwind first order scheme has been used for the faster convergence. Both the stationary and rotating domain are considered as a fluid domain. Air at 25°C is taken as a working fluid. Reference pressure for the simulation is set as 1atm. 160 rpm speed is applied to the rotating domain for downward flow simulation. Convergence criteria for CFD simulation is set as 0.001. A steady-state simulation is performed at each time step of co-simulation and CFD simulation converged after 259 iterations. Time taken in convergence is 30 minutes.

RESULTS AND DISCUSSION

Effect of the ceiling fan on thermal comfort index PMV, mean air temperature and cooling load is described in this section. Comparative analysis of the above mentioned parameter predicted by EnergyPlus model and co-simulation model is discussed.

Effect of ceiling fan on occupant's thermal comfort

PMV is taken as the thermal comfort index. PMV is a thermal sensation scale that ranges from cold (-3) to hot (+3), 0 refers the neutral sensation. PMV is developed by Fanger et al. 1970 and adopted by ISO standard. PMV can be calculated using equation 1. h_c (heat transfer coefficient) and t_a (mean air temperature) of equation 1 are dependent on the air velocity, hence PMV can be improved by increasing the air velocity through ceiling fan. In this study, PMV is calculated at each time step of co-simulation with the variables mean radiant temperature, relative humidity, clothing value and metabolic rate exported by EnergyPlus through external interface and air velocity and air temperature extracted from CFD simulation. Air velocity and air temperature are area weighted average of a virtual plane created at the height of 1m from the floor in the CFD post processing. 1m height is selected considering it as working height of the occupants. Figure 7 shows that PMV predicted by the co-simulation model pushed towards the cooler region than predicted by the EnergyPlus model. This is because of two reasons, first in co-simulation method mean air temperature is taken at height of 1m and extracted from CFD simulation whereas in EnergyPlus model mean air temperature is taken as well mixed average value of occupied zone which is higher than the previous one second in co-simulation method air flow induced by ceiling fan is considered which increases the convective heat transfer and in consequence lowers the mean air temperature and thermal comfort index. The cooling effect is observed more during morning and night because of the low mean air temperature predicted by co-simulation model as can be inferred from Figure 8. It can be concluded that air flow induced by ceiling fan improved the occupant's thermal comfort index.

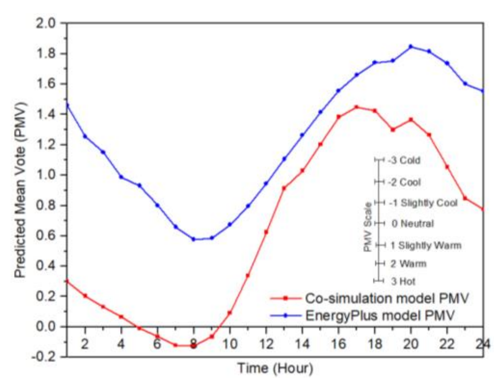

Figure 7 PMV comparison predicted by EnergyPlus co-simulation model

$$PMV = [0.303e^{-0.036M} + 0.028]\{(M - W) - 3.96E^{-8} f_{cl}[(t_{cl} + 273)^4 - (t_r + 273)^4] - f_{cl} h_c (t_{cl} - t_a) - 3.05[5.73 - 0.007(M - W) - p_a] - 0.42[(M - W) - 58.15] - 0.0173M(5.87 - p_a) - 0.0014M(34 - t_a)\} \quad (1)$$

Effect of ceiling fan on mean air temperature of the building

Mean air temperature of the building is predicted by EnergyPlus using nodal model. Nodal model is based on the fundamental of heat balance. It assumes different zones as separate nodes, and at each node, energy balance is performed according to equation 2. h_t (heat transfer coefficient) of equation 2 is determined by the empirical method in EnergyPlus model. In co-simulation model this h_t is determined by the CFD simulation and sent back to EnergyPlus for all the surfaces. Air flows induced by ceiling fan helps in improving the heat transfer coefficient and consequently mean air temperature is improved by co-simulation model. Figure 8 shows the comparison of mean air temperature predicted by EnergyPlus and co-simulation model. It can be observed from figure 8 that difference of mean air temperature predicted by both models ranges from 1.3 to 3.7 °C for the simulation period

$$C_z \frac{dT_z}{dt} = \sum_{i=1}^{N_s} Q_i + \sum_{i=1}^{N_{surface}} h_t A_i (T_{si} - T_z) + \sum_{i=1}^{N_{zones}} m_i C_p (T_{zi} - T_z) + m_{inf} C_p (T_\infty - T_z) + Q_{sys} \quad (2)$$

Where,

$C_z \frac{dT_z}{dt}$ energy change rate

$\sum_{i=1}^{N_s} Q_i$ sum of convective heat transfer through source or sink

$\sum_{i=1}^{N_{surface}} h_t A_i (T_{si} - T_z)$ sum of convective heat transfer from building envelope

$\sum_{i=1}^{N_{zones}} m_i C_p (T_{zi} - T_z)$ energy from neighboring zone and air mixing

$m_{inf} C_p (T_\infty - T_z)$ total energy of infiltration

Q_{sys} system output

Study of sensible cooling load

Due to the reduction of mean air temperature of the occupant's zone by co-simulation model sensible cooling load predicted by it is less than that predicted by the EnergyPlus model. Figure 9 presents the difference of sensible cooling load predicted by the co-simulation and EnergyPlus model. It is observed that co-simulation model reduces the sensible cooling load ranging from 20% to 45% than EnergyPlus model for the period of simulation.

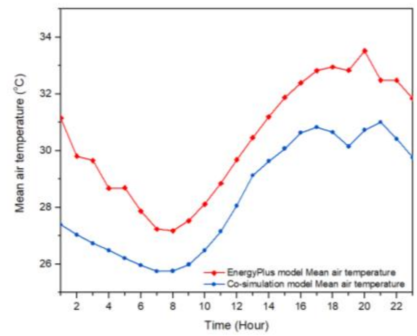

Figure 8 Mean air temperature comparison predicted by EnergyPlus co-simulation model

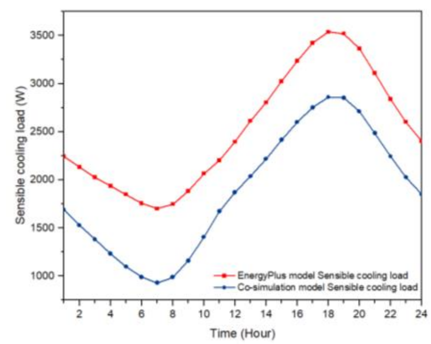

Figure 9 Sensible cooling load comparison predicted by EnergyPlus co-simulation model

CONCLUSIONS

A generalized interface is developed to couple nodal model and CFD. This interface is used to perform co-simulation between the two to incorporate the effect of the ceiling fan in whole building simulation tool, i.e., EnergyPlus to predict improved occupant's thermal comfort and associated reduction of cooling energy demand. Thermal comfort index PMV, mean air temperature and sensible cooling load predicted by EnergyPlus and co-simulation model are compared. It is found that PMV predicted by co-simulation model tends towards cooler region than predicted by EnergyPlus and average percent difference of PMV predicted by both the models for the period of simulation is approximately 38%. Similarly mean air temperature and sensible cooling load predicted by co-simulation model is less than that of the EnergyPlus model and average percent differences for the simulation period are approximately 6% and 27% respectively. It can be concluded that air flow induced by ceiling fan helps in improving the thermal comfort and associated cooling load reduction. Numerical results provide an insightful understanding of the usefulness of the method in modeling the strong air flow circulation and its effect on occupant's thermal comfort and building thermal performance.

The proposed method enables accurate estimation of the effect of ceiling fans in air-conditioned or naturally ventilated spaces. The method discussed in this study can be a useful resource to model the complexities of naturally ventilated or mixed-mode operated buildings to extend the thermal comfort envelope during summer for warm environments.

NOMENCLATURE

f_{cl}	Garment insulation factor $(m^2K/_{W})$
h_c	Heat transfer coefficient $(W/_{m^2K})$
I_{cl}	Resistance to sensible heat transfer
M	Metabolic rate $(W/_{m^2})$
T_{cl}	Cloth temperature (K)
T_r	Mean radiant temperature (K)
p_a	Vapor pressure of air (kPa)

REFERENCES

A. Inc. Ansys fluent documentation 2014

Chen, Q., Peng, X., Van Passen, A. 1995. Prediction of room thermal response by CFD technique with conjugate heat transfer and radiation models. ASHRAE Transactions 10:50-60

Crawley, D. 2001. EnergyPlus: creating a new-generation building energy simulation program. Energy and Buildings 2001:33:319-331

Djunaedy, E., Hensen, J., Loomans, M. 2003. Toward external coupling of building energy and airflow modeling programs. ASHRAE Transaction 109(2):771-87

Fanger, P. O. 1970. "Thermal Comfort Analysis and Applications in Environmental Engineering", Danish Technical Press, Copenhagen, Denmark

James, P.W., Sonne, J.K., Vieira, R.K., Parker, D.S., Anello, M.T. 1996. Are energy savings due to ceiling fans just hot air? ACEEE Summer Study on Energy Efficiency in Buildings. FSEC-PF-306-96

Morton-Gibson, M., Coutier, P.J., Place, W.J. 1985. Effects of fan velocity on thermal comfort in an office building, Proceedings of the 10th National Passive Solar Conference, the Solar Energy & Utilities Conference, the Daylighting Applications Conference, the Building with the Sun Conference 406–409

McIntyre, DA. Preferred air speed for comfort in warm conditions. ASHRAE Transactions 1978;84(2), 263--277Melikov A, Kaczmarczyk J. Air movement and perceived air quality. Building and Environment. 2012;47:400–9

Nagai, T., Kurabuchi, T. 2009. A two-node heat and mass transfer network model for long-term simulation of void space of high-rise apartment houses equipped with gas-fired boilers. In: 11th international IBPSA conference p. 434-41

Negrao, C. 1998. Integration of computational fluid dynamics with building thermal and mass flow simulation. Energy and Buildings 27:155-65

Rohles, F.H., Kons, S.A., Jones, B.W. 1983. Ceiling fans as extenders of the summer comfort envelope, ASHRAE Transactions 89 (part A1) 245–262

Scheatzle, D., H. Wu, and J. Yellot. 1989. Extending the summer comfort envelope with ceiling fans in hot, arid climates. ASHRAE Transactions 95(1)

Son H. Ho, Luis Rosario, Muhammad M. Rahman 2005. Effect of using ceiling fan on human thermal comfort in air-conditioned space, in: AIAA 5734, 3rd Intl. Energy Conversion

Wang, L., Wong, NH. 2008. Coupled simulations for naturally ventilated residential buildings. Automation in Construction 17:386-98

Wetter, M., Nouidui, T.S., Haves, P., Zhang, R., Li, Z. 2011. Building controls virtual test bed interface

Wong, N.H., Wang, L., Aida, N.C., Pandey, A.R., Wei X. 2005. Effects of double glazed façade on energy consumption, thermal comfort and condensation for typical office building in Singapore. Energy and Buildings 37:563-72

Zhai Y, Zhang Y, Zhang H, Pasut W, Arens E, Meng Q, Human comfort and perceived air quality in warm and humid environments with ceiling fans, Build. Environ. 90 (2015) 178–185

Zhang Y, Wang J, Chen H, Zhang J, Meng Q. Thermal comfort in naturally ventilated buildings in hot-humid area of China. Building and Environment 2010;45(11):2562–70

Zhai, Z., Chen, Q., Klems, J., Haves, P. 2001. Strategies for coupling energy simulations and computational fluid dynamics programs. In: Proceedings of 7th international IBPSA conference. p.59-66

Zhang, R., Lam, K.P., Yao, S., Zhang, Y. 2013. Coupled *EnergyPlus* and computational fluid dynamics simulation for natural ventilation. Building and Environment 68:100 -113