Annual Sessions of IESL, pp. [201 - 208] 2015

Prediction of Energy Performance of an Office Building through Combined Energy Simulation and Computational Fluid Dynamics

R. M. P. S. Bandara, R. A. Attalage and M. Vignarajah

Abstract: At present, high emphasize is given for designing energy efficient buildings in order to reduce their energy consumption and to limit the carbon footprint on the local and global environment. Energy Simulation is the most popular method in predicting the energy performance of buildings during the conceptual stage. However, it is observed that Energy Simulation tools show certain inherent deficiencies in predicting the energy performance of buildings. The said tools do not have the capacity to model air circulation through the building space explicitly. Energy Simulation tools mainly rely on the simplifying assumption that air within a thermal zone of a building is *well-mixed*. Furthermore, convective heat transfer coefficients of building surfaces are calculated using set empirical correlations. The literature also reveals that most Energy Simulation tools under-predict energy consumption in buildings, especially located in tropical regions. On the other hand, Computational Fluid Dynamics tools are capable of predicting the indoor flow field comprehensively. On this basis, the paper explains how Energy Simulation can be coupled with Computational Fluid Dynamics in predicting the energy performance of a building more accurately through complementary data exchange between the tools. The office building considered in the study is to be constructed in the suburbs of Colombo. The analysis uses *EnergyPlus 8.0* and *Ansys Fluent 6.3* as the tools for conducting Energy Simulation and Computational Fluid Dynamics respectively. The study shows that the coupled scheme predicts a considerably higher annual energy consumption of the building compared to that given by conventional Energy Simulation using *EnergyPlus*.

Keywords: Energy Performance, Energy Simulation, Computational Fluid Dynamics

1. Introduction

Buildings account for nearly 40% of the global energy consumption, 16% of the world's fresh water, 25% of the forest timber while emitting almost 70% of oxides of sulphur and 50% of carbon dioxide gas annually [1]. Hence at present, high emphasize is given for designing energy efficient buildings in order to reduce their energy consumption and to limit the carbon footprint on the local and global environment.

A building is a complex system with multiple interacting physical processes taking place simultaneously. Performance of buildings can be analysed based on the following criteria:

- Energy performance
- Indoor environment for human comfort and health
- Environmental degradation
- Economic aspects

Building energy performance analysis, during the conceptual stage, is mostly done through Energy Simulation (ES). This is an approach that analyses thermal aspects, day-lighting, moisture, acoustics, airflow and indoor air quality of buildings [2]. A whole building energy simulation tool such as *EnergyPlus* [3] serves this purpose. Energy Simulation is based on the principles of energy and mass conservation. Inputs for the process mainly consists of the building geometry, weather data, Heating, Ventilation and Air Conditioning (HVAC) systems and components, internal loads, operating strategies and schedules and simulation specific parameters. ES tools are capable of predicting space-averaged indoor conditions, cooling/heating loads and energy consumption etc on an hourly or sub-hourly basis for a particular design day, a specific time

***Eng. R. M. P. S. Bandara**, B.Sc. Eng. (Moratuwa), M.Eng. (Moratuwa), AMIE(Sri Lanka), MSLAAS(SL), Senior Lecturer, Department of Mechanical Engineering, General Sir John Kotelawala Defence University, Sri Lanka*
***Eng. (Prof.) R. A. Attalage**, B.Sc. Eng. (Moratuwa), M.Eng. (AIT), DEA (Paris), PhD (Paris), C.Eng., MIE(Sri Lanka), Senior Professor, Department of Mechanical Engineering, University of Moratuwa, Sri Lanka*
***Eng. M. Vignarajah**, B.Sc. Eng. (Peradeniya), MPhil. (Peradeniya), M.Sc.(USA), MASCE(USA), MSLAAS(SL), Senior Lecturer, Department of Civil Engineering, General Sir John Kotelawala Defence University, Sri Lanka*

period or for a reference year or more. However, it is observed that Energy Simulation tools show certain inherent deficiencies in predicting the energy performance of buildings. ES tools assume uniform air temperature within the thermal zone due to the application of the "well-mixed" model. This assumption may be appropriate for small buildings. However, for moderate and large buildings, those typically produce non-uniform air temperature distributions within the occupied space, such as displacement ventilation systems, the said tools cannot accurately predict the energy consumption [4]. Furthermore, convective heat transfer coefficients utilized by the Energy Simulation tools are generally determined through empirical correlations and have limited applicability. They are unable to provide information on the airflow field introduced by building spatial configurations especially through natural ventilation [4]. ES tools do not have the capacity to model air circulation through the building space explicitly. Knowledge on the airflow field is vital in predicting the temperature field of building air and heating and/or cooling load and hence the energy consumption. Also spatially-averaged thermal comfort predictions are not sufficient to satisfy advanced design requirements at present [4]. Many Energy Simulation tools under-predict energy consumption in buildings [5]. Some studies [6, 7] suggest that this discrepancy of energy consumption may even reach up to 37%.

On the other hand, Computational Fluid Dynamics (CFD) tools can predict airflow paths, velocities, relative humidity and contaminant concentrations within an occupied space of a building extensively and accurately. Also, they are capable of determining the temperature distribution in the building space and convective heat transfer coefficients of the building envelope. The predictions can be further extended to determine thermal comfort indices such as Predicted Mean Vote (PMV), Percentage of People Dissatisfied (PPD) due to discomfort, Percentage Dissatisfied (PD) due to draft and ventilation effectiveness [4]. For CFD, boundary of the solution domain is the inside surface of the building. Hence, it is difficult to predict the corresponding boundary conditions for CFD simulations since they depend on many parameters such as construction details of the building envelope, outside weather conditions etc. However, this information is readily available with ES tools.

On this basis, it is clear that if Energy Simulation and Computational Fluid Dynamics are combined together, more accurate predictions for building energy consumption can be made through complementary data exchange between the said tools. The paper discusses how this concept is applied to an office building to be constructed at Ratmalana in predicting its energy consumption more accurately.

2. Governing Principles

2.1 Energy Simulation

Energy balance equations for building zone air and surface heat transfer are two essential equations that an ES tool should solve [4]. The energy balance equation for building air is in the form [4]:

$$\sum_{i=1}^{N} q_{i,c}A_i + Q_{\text{other}} - Q_{\text{heat\_extraction}} = \frac{\rho V_{\text{building}} C_p \Delta T}{\Delta t} \quad \text{....(1)}$$

where

$\sum_{i=1}^{N} q_{i,c}A_i$ = Convective heat transfer from enclosure surfaces to building air

$q_{i,c}$ = Convective flux from surface i

N = Number of enclosure surfaces

A_i = Area of surface i

Q_{other} = Heat gains from lighting, occupants, appliances, infiltration etc.

$Q_{heat\_extraction}$ = Heat extraction rate of the building

$\frac{\rho V_{building} C_p \Delta T}{\Delta t}$ = Rate of change of energy in building air

ρ = Air density

$V_{building}$ = Volume of building

C_p = Specific heat capacity of air

ΔT = Change of building air temperature

Δt = Sampling time interval

When the building air temperature is kept constant (ΔT = 0) heat extraction rate is equal to the cooling and/or heating load.

The convective heat fluxes are determined from the energy balance equations for the corresponding surfaces as shown in Figure 1. A similar energy balance is performed for each surface. The surface energy balance equation can be written as [4]:

$$q_i + q_{ir} = \sum_{k=1}^{N} q_{ik} + q_{i,c} \qquad \text{....(2)}$$

where

q_i = Conductive heat flux on surface i

q_{ir} = Radiative heat flux from internal heat sources and solar radiation

q_{ik} = Radiative heat flux from surface i to surface k

q_{ic} = Convective heat flux from surface i

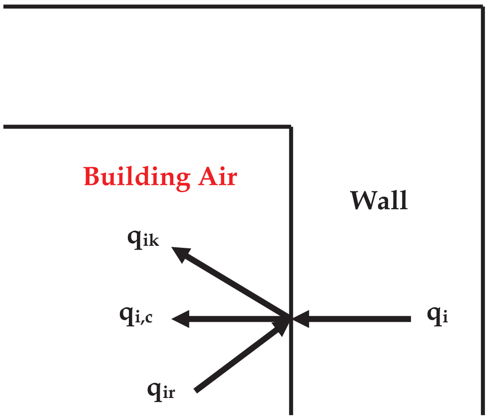

Figure 1 - Energy Balance of Surfaces in a Building [4]

q_i can be determined by transfer functions or by solving the discretized heat conduction equations of the surfaces using finite difference schemes.

Convective heat flux is determined by [4]:

$$q_{i,c} = h_c\left(T_i - T_{\text{building}}\right) \qquad \text{....(3)}$$

where

h_c = Convective heat transfer coefficient

T_{building} = Building air temperature

T_i = Building surface temperature

The convective heat transfer coefficient, h_c, is not known. ES tools such as *EnergyPlus* estimate h_c by empirical equations or as constants.

The detailed natural convection model in *EnergyPlus* correlates the convective heat transfer coefficient (h_c) to the surface orientation and the temperature difference between the surface and zone air (ΔT) as follows [8]:

- If $\Delta T = 0$ or a vertical surface, then

$$h_c = 1.31\|\Delta T\|^{1/3} \qquad \text{....(4)}$$

- If $\Delta T < 0$ with an upward facing surface or $\Delta T > 0$ with an downward facing surface, then

$$h_c = \frac{9.482\|\Delta T\|^{1/3}}{7.283 - \|\cos\Sigma\|} \qquad \text{....(5)}$$

where Σ is the surface tilt angle.

- If $\Delta T > 0$ with an upward facing surface or $\Delta T < 0$ with an downward facing surface, then

$$h_c = \frac{1.810\|\Delta T\|^{1/3}}{1.382 - \|\cos\Sigma\|} \qquad \text{....(6)}$$

The simple natural convection model uses constant coefficients for different configurations as given in Table 1.

Table 1 - Simple Natural Convection Model [8]

Configuration	h_c (W/m²K)
Horizontal surface with reduced convection	0.948
Horizontal surface with enhanced convection	4.040
Vertical surface	3.076
Tilted surface with reduced convection	2.281
Tilted surface with enhanced convection	3.870

The ceiling diffuser model in *EnergyPlus* correlates convective heat transfer coefficient to the supply mass flow rate (ACH) as shown in Table 2.

Table 2 - Ceiling Diffuser Model [8]

Building Element	h_c (W/m²K)
Floors	$3.873 + 0.082(ACH)^{0.980}$
Ceilings	$2.234 + 4.099(ACH)^{0.503}$
Walls	$1.208 + 1.012(ACH)^{0.604}$

If the building air temperature, T_{building}, is assumed to be uniform and known, the interior surface temperatures, T_i, is determined by simultaneously solving the surface heat balance equation (2). Building thermal load is then calculated from the convective heat transfer from enclosure surfaces using equation (1).

2.2 Computational Fluid Dynamics

In Computational Fluid Dynamics numerical techniques are applied for solving Navier-Stokes equations for fluid flow and heat transfer. Navier-Stokes equations are derived through the application of the conservation laws of mass (continuity), momentum and energy (first law of thermodynamics) to a control volume of fluid. In addition to the aforementioned basic set of governing equations, different models such as turbulence, radiation, combustion etc. may be incorporated depending on the problem being handled. The general form of the governing equations takes the following form [4]:

$$\frac{\partial \phi}{\partial t} + (\boldsymbol{V} \cdot \boldsymbol{\nabla})\, \phi - \Gamma_\phi \nabla^2 \phi = S_\phi \qquad \text{....(7)}$$

where
t = Time
φ = General variable
V = Velocity vector
Γ_φ = Diffusion coefficient
S_φ = Source term

The governing equations in CFD are highly non-linear in nature. Hence, they are solved by discretizing the equations using finite volume methods converting them to a set of algebraic equations. The spatial domain is divided into a finite number of discrete cells (or nodes) creating a computational mesh of acceptable resolution. Appropriate boundary conditions are assigned for the computational domain depending on the problem being handled. All transport equations are solved at each node point of the mesh at each time step through an iterative process until the solution meets a preset convergence criterion.

The accuracy of the CFD solution is highly sensitive to the boundary conditions assigned for the domain. Hence, in modelling indoor flows in buildings, boundary conditions related to air supply, air exhaust, envelope surfaces and internal objects highly influence the CFD solution. On this basis, supply air temperature, velocity and level of turbulence comprise the inlet boundary conditions. The interior surface temperatures and/or heat fluxes of the building envelope establishes the vital thermal boundary conditions for the problem.

3. Coupling Approach

Many attempts have been made for coupling Energy Simulation and Computational Fluid Dynamics mainly since 1990. Negrao [9] performed a complete iterative coupling between ES and CFD. A full iterative strategy was implemented, where coupled variables were exchanged at each iterative step until a convergence criterion was reached at each time step. Beausoleil-Morrison [10, 11, 12] continued the work of Negrao [9, 13] with the investigation of the coupling between ES and CFD. A conflation controller was established to configure the CFD model at each time step. Bartak et al. [14] conducted an empirical validation of the coupled model of Beausoleil-Morrison [11]. Djunaedy et al. [15, 16], Chen et al. [17] and Zhai et al. [4] analysed the pros and cons of internal coupling of the ES and CFD. Zhai et al. [18, 19, 20] investigated the different coupling strategies extensively. Their results revealed that for rooms of moderate size, without significant temperature stratification, coupling of ES and CFD gives marginal improvement in energy performance predictions. However, those with large temperature stratification, the discrepancy between the coupled approach and ES alone can be as high as 42%. Wang [21] and Wang and Chen [22] proved that the combined ES and CFD approach has a unique solution. Wang and Wong [23] developed a text-based interface for automated coupling for exchanging information between TAS (ES tool) and Fluent (CFD tool). According to Djunaedy [24] coupling between ES and CFD is categorized as follows:

- Internal coupling (Hard coupling) - Two or more sets of equations are combined and solved at the same time (Conjugate heat transfer method)
- Internal coupling (Loose coupling) - Two or more sets of equations are solved separately, and exchange data during calculation
- External coupling (Loose coupling) - Two or more set of equations solved separately, in ES and CFD programs, and exchange data during calculation

The application of the conjugate heat transfer approach has several disadvantages. The difference in stiffness of the fluid and the solid side of the model leads to difficulties in obtaining a converged solution [25]. It is computationally expensive since the computing time increases drastically due to the difference in the time scale related to dynamics in fluids (few seconds) and dynamics in solids (few hours) encountered in buildings [4]. Although Internal coupling (Loose coupling) solves some of the issues in the first method, internal coupling approach as a whole is a computationally expensive approach [24].

Benefits of external coupling include [26]:

- Computationally less expensive
- ES and CFD models can be maintained and updated individually

Moreover, building simulation research [27] reveals that the difference in results between internal and external coupling is not significant.

4. Methodology and Approach

4.1 Energy Simulation Tool

EnergyPlus [3] v. 8.0 is used as the ES tool during present work. It is a new generation building energy modelling tool based on *DOE* (U.S. Department of Energy) – *2* and *BLAST* (Building Loads Analysis and System Thermodynamics), with numerous capabilities and was first released in 2001. It can model heating, cooling, lighting, ventilation, other energy flows, water usage etc. in buildings and includes many innovative simulation capabilities [3].

4.2 Computational Fluid Dynamics Tool

Ansys Fluent v. 6.3 [28] is used as the CFD tool for the study. *Fluent* is one of the most widely used and extensively validated software available for flow modelling. It is capable of modelling turbulence, heat and mass transfer, combustion, multi-phase flow etc.

4.3 Coupling Platform

MATLAB R2012a forms the coupling platform for ES and CFD tools. It is a high-level language and interactive environment for numerical computation, visualization, and programming [29]. *MATLAB* utilizes a comprehensive collection of toolboxes that extend its potential to solve a variety of problems.

4.4 ES Problem Setup

The single-storey office building under consideration has overall dimensions of 8.0 m x 6.0 m x 3.5 m. ES computational model of the building was created using *Google SketchUp* v. 8.0 with *OpenStudio* plug-in v. 1.0.7 and is shown in Figure 2. It consists of a single thermal zone. The computational model of the building is generated using inputs through both *Google SketchUp* and *IDF Editor* of *EnergyPlus*. Weather information was incorporated to the model through the .epw file available for the Ratmalana area. Tables 3 and 4 give the details of thermal and electrical loads and construction details of the building respectively.

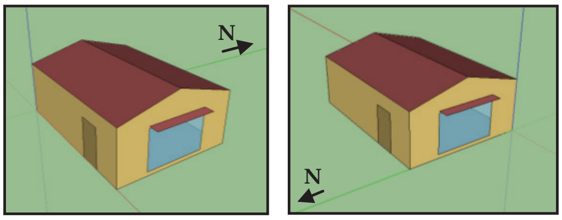

Figure 2 – Computational Model of Office Building for Energy Simulation

Table 3 - Thermal and Electrical Loads

Load / System	Rating and Description
Occupancy	10 nos. of occupants involved in general office work with the specified occupancy schedule.
HVAC system	Temperature control through dual set point, where 20 ^{0}C for heating and 25 ^{0}C for cooling effective from 7.00 to 19.00 hrs. Maximum indoor air velocity is 0.2 ms^{-1}.
Artificial lighting	200 W
Electrical equipment rating	500 W
Building lighting control mechanism	Continuously dims artificial lights to match an illumination set point of 500 lx at two reference points at a working plane of 0.8 m above the floor level, with the variation of day light.

Table 4 – Construction Details

Element	Construction Details
Walls	9" thick brickwork
Roof	Pitched roof of 15^0 with 25 mm thick Calicut tiles
Floor	10 mm thick ceramic tiles on a 150 mm thick reinforced concrete slab.
Doors	Each of 1.1 m x 2.0 m, made of plywood.
Windows	3.0 m x 2.0 m double pane windows with 4 mm thick glass and 2 mm thick air space. There exists 0.2 m of wall below the window and 0.5 m of wall above the window. The edge of each window is located 1.5 m from the respective wall edge.
Shading Overhangs	Depth 0.5 m with 0.1 m height above the window. Tilt angle is 90^0.

4.5 CFD Problem Setup

The CFD model of the building was created using the solid modelling software *GAMBIT* v. 2.2 [30] and is shown in Figure 3. It consists of 686789 tetrahedral/hybrid mesh volumes in the computational domain. Separate boundary meshes were created for each solid surface of the building.

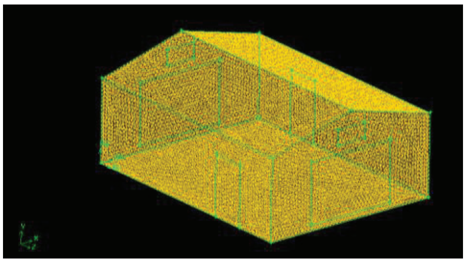

Figure 3 – Computational Model for CFD

Table 5 gives the modelling parameters for the CFD simulation.

Table 5 – CFD Modelling Parameters

Parameter	Model/Value
Building air supply	Inlet duct on west wall Discharge on east wall
Supply air temperature	16 ^{0}C
Turbulence Model	k-ε RNG
Discretization scheme	QUICK
Near-wall Treatment	Enhanced Wall Treatment
Pressure-Velocity Coupling	Coupled

4.6 Simulation Setup

External coupling (Loose coupling) approach is adopted for combining ES and CFD on *MATLAB* platform in the present study. Hence, individual simulation tools achieve the status of convergence before exchanging the variables between them. Internal surface temperatures of the building (T_w) generated by *EnergyPlus* and surface convective heat transfer coefficients (h_c) predicted by *Ansys Fluent* are the exchange variables for the coupled simulation.

Since execution of the coupled simulation to predict annual energy consumption of the building is highly computationally expensive, simulations are only conducted for the following cases with respect to the weather file for Ratmalana area:

- Case 1 - Day recording the maximum dry bulb temperature
- Case 2 - Day recording the minimum dry bulb temperature

Time step for complimentary data exchange is taken as 1 hour to compensate between computational cost and accuracy of the solution. Workflow of the coupled simulation is shown in Figure 4. Final ES solution provides the building energy consumption for the particular case.

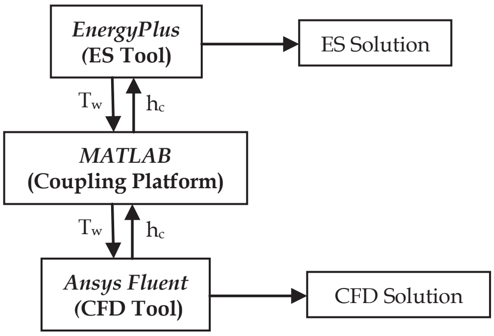

Figure 4 – Workflow of Coupled Simulation

5. Simulation Results

The combined ES and CFD setup was run on an Intel Core i5 3.2 GHz workstation of 4.0 GB RAM. It took 13 hours and 40 minutes for each simulation to complete. Figures 5 and 6 illustrate the daily variation of inside surface temperatures of the building predicted by *EnergyPlus* for Cases 1 and 2.

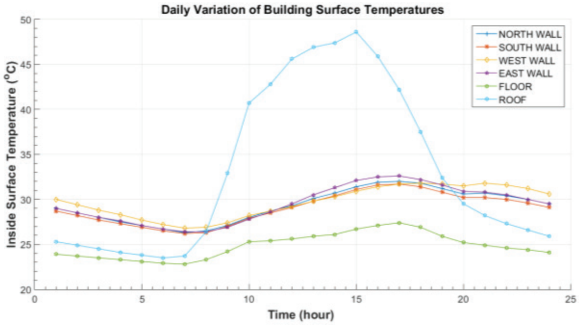

Figure 5 – Daily Variation of Inside Surface Temperature – Case 1

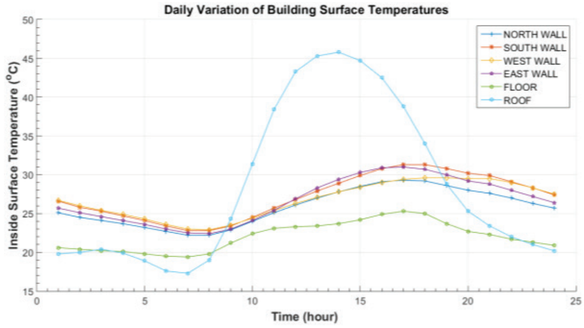

Figure 6 – Daily Variation of Inside Surface Temperature – Case 2

Figures 7 and 8 show the daily variation of wall convective heat transfer coefficient related to ES and coupled approach for Cases 1 and 2.

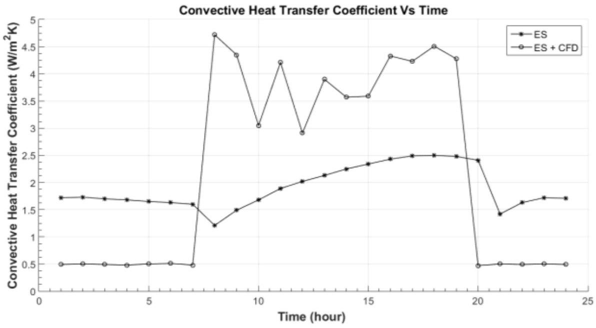

Figure 7 – Daily Variation of North Wall Convective Heat Transfer Coefficient - Case 1

Figures 9 and 10 illustrate the velocity flow fields on the mid plane of the building predicted by the CFD tool.

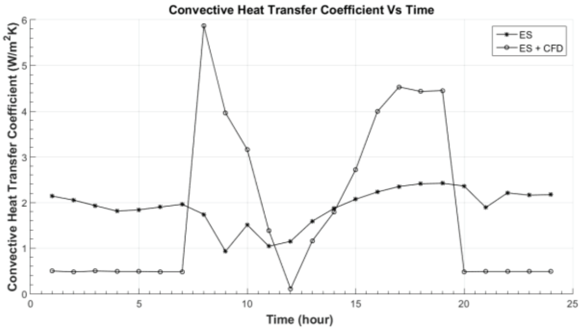

Figure 8 – Daily Variation of South Wall Convective Heat Transfer Coefficient - Case 2

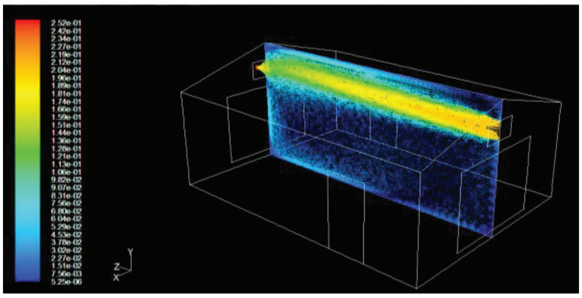

Figure 9 – Predicted velocity flow field when HVAC system is in operation

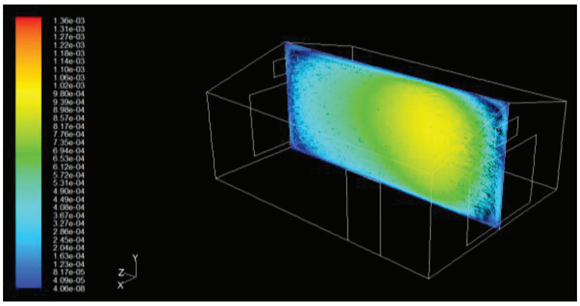

Figure 10 – Predicted velocity flow field when HVAC system is not in operation

Table 6 gives a comparison of the building energy consumption related to the two approaches for Case 1 and Case 2.

Table 6 – Building Energy Consumption

Case	Energy Consumption (MJ)		Discrepancy (%)
	ES only	ES + CFD	
1	258.66	310.52	20.0
2	182.24	213.42	17.1

According to Table 6, it is observed that Case 1 and Case 2 report a discrepancy of 20% and 17.1% respectively.

In the proposed model, the respective convective heat transfer coefficients were calculated based on the temperature of a point located just outside the thermal boundary layer of the corresponding surface of the building. Hence, the value of the convective heat transfer coefficient is strongly dependent on the location of this point. Also, the influence due to movement of occupancy was not considered during the analysis.

6. Conclusion

The study considered only two cases for the analysis since it is not feasible to perform the coupled simulation for the entire 365 days of the year due to the high computational cost involved. It is seen that for both cases (this is the sample) considered, the discrepancy related to energy consumption between ES and coupled approach is significant. This is in good agreement with the previous studies conducted by different researchers. Hence, it is reasonable to expect that the same trend prevails for the entire year (this is the population). However, this needs to be further justified through an appropriate statistical approach.

It is essential to analyse whether similar results are obtained for different air supply configurations of the building. Furthermore, validation of the model needs to be conducted through appropriate measurements. This will be the next step of the present effort.

References

1. Ghiaus, C. & Inard, C., *Energy and environmental issues of smart buildings, A Handbook for Intelligent Building* [Online], 2004. Available from: http://www.ibuilding.gr/handbook/, Visited, 15th March 2015.

2. Mumovic, D., & Santamouris, M., *A Handbook of Sustainable Building Design and Engineering: An Integrated Approach to Energy, Health and Operational Performance*, Earthscan, 2009.

3. Crawley, D., et al. 2001 "*EnergyPlus*: creating a new-generation building energy simulation program", *Energy and Buildings*, Vol. 33, 2001, pp. 319-331.

4. Zhai, Z., & Chen, Q., "Strategies for Coupling Energy Simulation and Computational Fluid Dynamics Programs", *Proceedings of the 7th International IBPSA Conference*, Rio de Janeiro, August 13-15, 2001.

5. Lomas, K. J., Eppel, H., Martin, C., & Bloomfield, D., "Empirical validation of thermal building simulation programs using test room data", vols. 1–2: Final Report, *IEA Energy Conservation in Buildings and Community Systems Programme* Annex. 21 and *IEA Solar Heating and Cooling Programme* Task 12, 1994 (IEA 1994).

6. Spitler, J. D., Pedersen, C. O., & Fisher, D. E., "Interior Convective Heat Transfer in Buildings with Large Ventilative Flow Rates", *ASHRAE Transactions*, Vol. 97, 1991, pp. 505-515.

7. Lomas, K. J., "The U.K. Applicability Study: An Evaluation of Thermal Simulation Programs for Passive Solar House Design", *Building and Environment*, Vol. 31, No. 3, 1996, pp. 197-206.

8. *EnergyPlus* Engineering Reference, University of Illinois & Ernest Orlando Lawrence Berkeley National Laboratory, September 2014.

9. Negrao, C.O.R., "Conflation of Computational Fluid Dynamics and Building Thermal Simulation", *PhD Thesis*, University of Strathclyde, 1995.

10. Beausoleil-Morrison, I., & Clarke, J.A., "The Implications of using the Standard k-ε Turbulence Model to Simulate Room Air Flows which are not fully Turbulent", *Proceedings of* ROOMVENT, 1998, pp. 99-106.

11. Beausoleil-Morrison, I., "The Adaptive Coupling of Heat and Air Flow Modelling Within Dynamic Whole-Building Simulation", *PhD Thesis*, Energy Systems Research Unit, Department of Mechanical Engineering, University of Strathclyde, Glasgow, UK, 2000.

12. Beausoleil-Morrison, I., Clarke, J. A., Denev, J. A., Melikov, A., & Stankov, P., "Further Developments in the Conflation of CFD and Building Simulation", *Proceedings of the 7th International IBPSA Conference*, Rio de Janeiro, Brazil, 2001, pp. 1267-1274.

13. Negrao, C.O.R., "Integration of Computational Fluid Dynamics with building Thermal and Mass flow Simulation", *Energy and Buildings*, Vol. 27, 1998, pp. 155-165.

14. Bartak, M., et al, "Integrating CFD and Building Simulation", *Building and Environment,* Vol. 37, 2002, pp. 865-871.

15. Djunaedy, E., Hensen, J. L. M., & Loomans, M. G. L. C., "Towards External Coupling of Building Energy and Airflow Modelling Programs", *ASHRAE Transactions*, Vol. 109(2), 2003, pp. 771-787.

16. Djunaedy, E., Hensen, J. L. M., & Loomans, M. G. L. C., "Selecting an Appropriate Tool for Airflow Simulation in Buildings", *Building Services Engineering Research and Technology*, Vol. 25(3), 2004, pp. 289-298.

17. Chen, Q., Peng, X., & Van Passen, A. H. C., "Prediction of Room Thermal Response by CFD Technique with Conjugate Heat Transfer and Radiation Models", *ASHRAE Transactions*, Vol. 101, 2003, pp. 50-60.

18. Zhai, Z., et al, "On Approaches to Couple Energy Simulation and Computational Fluid Dynamics Programs", *Building and Environment*, Vol. 37, No. 8, 2002, pp. 857-864.

19. Zhai, Z., et al, "Solution Characters of Iterative Coupling between Energy Simulation and CFD Programs", *Energy and Buildings*, Vol. 35(5), 2003, pp. 493-505.

20. Zhai, Z., et al, "Pressure Boundary Conditions in A Multi-Zone and CFD Program Coupling", *Proceedings of the 1st IBPSA-USA Conference*, Boulder, Colorado, USA, 2004.

21. Wang, L., "Coupling of Multi-Zone and CFD Programs for Building Airflow Distribution and Contaminant Transport Simulations", *PhD Thesis*, Purdue University, 2007.

22. Wang, L. Z., & Chen, Q. Y., "On Solution Characteristics of Coupling of Multi-Zone and CFD Programs in Building Air Distribution Simulation", *9th International IBPSA Conference*, Montreal, Canada, August, 2005.

23. Wang, L., & Wong, N. H., "Coupled Simulations for Naturally Ventilated Residential Buildings", *Automation in Construction,* Vo. 17, 2008, pp.386-398.

24. Djunaedy, E., "External Coupling between Building Energy Simulation and Computational Fluid Dynamics", *PhD Thesis*, The Eindhoven University of Technology, Netherlands, 2005.

25. Chen, Q., Peng, X., Paassen, A. H. C. V., "Prediction of Room Thermal Response by CFD Technique with Conjugate Heat Transfer and Radiation Models", *ASHRAE Transactions*, Vol. 101, No. 2, 1995, pp. 50-60.

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

27. Djunaedy, E., Hensen, J. L. M., & Loomans, M.G.L.C., "Comparing Internal and External Run Time Coupling of CFD and Building Energy Simulation Software", *Proceedings of the 9th ROOMVENT International Conference on Air Distribution in Rooms*, Coimbra, 2004, pp. 393 - 396.

28. *Ansys Fluent* v.6.3, Fluent Inc., 2006.

29. The MathWorks, *MATLAB* program v. 2012b. Natick, MA: The MathWorks, Inc., 2012.

30. *GAMBIT* v. 2.2, Fluent Inc., 2006.