รายงานวิจัยฉบับสมบูรณ์

โครงการ การวิเคราะห์ประสิทธิภาพของการใช้โปรแกรมร่วม
ของ Energy Simulation และ CFD เพื่อการประเมินผลและ
วิเคราะห์ความต้องการการใช้พลังงานและผลกระทบของ
สิ่งแวดล้อมภายในอาคารในขั้นตอนเบื้องต้นของการ
ออกแบบอาคาร

โดย ผศ.ดร. พฤทธิพงษ์ ไทยแช่ม

ธันวาคม 2552

สัญญาเลขที่ MRG4980164

รายงานวิจัยฉบับสมบูรณ์

โครงการ การวิเคราะห์ประสิทธิภาพของการใช้โปรแกรมร่วมของ Energy Simulation และ CFD เพื่อการประเมินผลและวิเคราะห์ความต้องการการใช้พลังงานและผลกระทบของสิ่งแวดล้อมภายในอาคารในขั้นตอนเบื้องต้นของการออกแบบอาคาร

โดย ผศ.ดร. พฤทธิพงษ์ ไทยแช่ม
มหาวิทยาลัย ชินวัตร

สนับสนุนโดยสำนักงานกองทุนสนับสนุนการวิจัย

รูปแบบ Abstract (บทคัดย่อ)

Project Code : MRG4980164

Project Title : Measurement and prediction of indoor environment by coupled building energy and CFD simulation

ชื่อโครงการ : การวิเคราะห์ประสิทธิภาพของการใช้โปรแกรมร่วมของ Energy Simulation และ CFD เพื่อการประเมินผลและวิเคราะห์ความต้องการการใช้พลังงานและผลกระทบของสิ่งแวดล้อมภายในอาคารในขั้นตอนเบื้องต้นของการออกแบบอาคาร

Investigator : Asst.Prof.Dr. Pruitipong Thaicham

ชื่อนักวิจัย : ผศ.ดร. พฤทธิพงษ์ ไทยแช่ม

E-mail Address : pruitipong@siu.ac.th

Project Period : 2yr 5mth

Abstract

The information provided by computer fluid dynamics (CFD) and building simulation (BS) program is essential for the evaluation of the most significant building performance, including thermal comfort, indoor air quality, mechanical system efficiency, and energy consumption. However, the uniform indoor air temperature assumed by ES simulations is not true for most indoor spaces. The assumption may cause serious problems in predicting building cooling/heating load for cases with air temperature stratification. The integration of these two applications can eliminate the primary assumptions employed in the separate simulations and thus results in more accurate predictions of building performance. For example, an ES program can provide building cooling load and interior surface temperatures of building envelopes to CFD as boundary conditions while CFD can determine surface convective heat fluxes for ES. This study describes an approach to integrate ES and CFD program and validates the coupled application by using sets of experimental data from literatures. The program was then used to calculate environmental indoor conditions of Shinawatra University's atrium, Thailand. The comparison of the simulated results with the experimental data reveals the benefits of the coupled simulation over the separated ES and CFD application.

Keywords : Coupled building simulation; Computational fluid dynamics; Energy simulation

1. Introduction

Computational fluid dynamics (CFD) and building simulation (BS) programs are widely used in building design processes. Building simulation mainly uses to model heat transfer in buildings based on heat balance methods while CFD can predict detailed airflow for indoor and outdoor. Detail information provided by CFD is such as distributions of air velocity, air pressure, temperatures, and turbulence by numerically solving the governing conservation equations of fluid flows. It is a reliable tool for the evaluation of thermal environment and contaminant distributions. However, due to long computational time and excessive computer resource requirement, the application of CFD for indoor environmental prediction has been limited. BS programs basically include two fundamental calculations: thermal simulation and airflow network to solve the heat and mass transfer and airflow in buildings.

Both BS and CFD tools greatly facilitate energy efficient sustainable building designs by providing predictions of thermal behaviors, indoor airflow of buildings and better understanding of the consequences of various design decisions. However, because BS assumes well mixed uniform indoor air, it can only provide the uniform results, which normally does not meet the requirements for detailed indoor environment analyses. Thus, advanced evaluation of building designs for thermal comfort can be achieved by complementary of these two programs. The coupled simulation of CFD and BS programs can eliminated a few assumptions employ in the separate applications, significantly reduce computation time of CFD, and result in accurate and quick predictions of building performance. For example, the boundary conditions of CFD are usually assumed with limited consideration for the thermal storage effects of walls, external conditions and interactions with building service systems. ES calculates its energy prediction based on a well-mixed assumption so that the definition of the convective heat transfer coefficient (CHTC) cannot capture the dynamics of the flow near the surfaces. The coupling between CFD and ES program is seen as an alternative to achieve better results because the two can provide boundary conditions to each other. ES program can provide building cooling load and interior surface temperatures as boundary conditions to CFD while CFD can determine surface convective heat fluxes for ES.

Negrao (1998) investigated the integration of the internal CFD module built into ESP-r simulation to perform the detailed cooling and heating load calculations. ESP-r calculated thermal boundary conditions such as inlet air temperature and velocity, including surface heat flux for CFD calculation. CFD then used this information to predict the detailed air temperature, air velocity and surface convection coefficients for ESP-r. The simulation was completed once the ESP-r calculated the detailed heating and cooling loads using the surface convection coefficients from CFD which showed promising results. Srebric *et al*. (2000) studied a coupled CFD simulation with the BS program called ACCURACY, which hourly calculated heating and cooling loads based on the energy balance method. ACCURACY predicted the wall surface temperatures and supplied air velocity based on the cooling load requirement. CFD then used those results to calculate the convection coefficients for ACCURACY. Beausoleil (2002) proposed the adaptive conflation of CFD with dynamic whole-building thermal simulation to improve the accuracy of the heat convective coefficients. The integration between CFD and ESP-r for air-conditioned rooms was improved by adding zero-equation turbulence model and wall functions in the built in CFD simulation. Zhai *et al*. (2003) developed several coupling methods to integrate EnergyPlus and CFD simulation (MIT-CFD). Energy simulation program could provide internal wall temperatures and energy loads of buildings to CFD as boundary conditions while CFD predicted convective heat transfer coefficient to accurately calculate energy consumption using EnergyPlus. Tan *et al*. (2005) studied external coupling for airflow simulation between PHOENICS and multi zone model program, MultiVent, and found that this integration can predict the wind buoyancy ventilation. However, the coupled program cannot accurately estimate wind-driven natural ventilation. Good accuracy of the integration could only be achieved with buoyancy effects for natural ventilation. Cheong *et al*. (2003) found that external coupled simulation between ESP-r thermal simulation with commercial CFD software FLUENT could provide the results as good as the internal coupled simulation.

To calculate heat transfers in buildings several BS programs have been internally or externally coupled with CFD simulation. Internal coupling can also be seen as subroutinazation. External coupling method is more flexible in choosing CFD turbulence models and can also minimize the source code changes by made use of existing packages in different domain, i.e. the thermal domain for energy simulation and the flow domain for CFD, and provides a mechanism for these programs to communicate. This study develops the external coupling implementation between BS and CFD simulation to improve the accuracy in assessing the performance for indoor environment of buildings. The results report validations of the program by experimental data from the literature. The coupled application was later used to determine temperature profiles of a large scale atrium of Shinawatra University, Thailand, which is compared with measurements.

2. Strategies and methods

2.1 Energy simulation principles

Two essential equations used by many ES programs are energy balance equations for room air and surface heat transfer. The energy balance equation for room air is

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

Where

$\sum_{i=1}^{N} q_{i,c} A_i$ is the convective heat transfer from enclosure surfaces to room air, $q_{i,c}$ is the convective heat flux from surface *i*, *N* is the number of enclosure surfaces, A_i is the area of surface *i*, Q_{other} is heat gains from lights, people, appliances, infiltration, etc., $Q_{heat\_extraction}$ is the heat extraction rate of the room, $\rho V_{room} C_p \Delta T/\Delta t$ is the energy change in room air. ρ is the air density, V_{room} is the room volume, C_p is the specific heat of air, ΔT is the temperature change of room air, and Δt is the sampling time interval, normally 1 hr. The heat extraction rate is the heating and cooling load when the room air temperature is maintained as constant ($\Delta T=0$).

The convective heat transfer from walls and windows is determined from the energy balance equation for that surface. The energy balance equation for a surface can be written as

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

where q_i is the conductive heat flux on surface *i*, which can be determined by transfer functions, by weighting factors, or by solutions of the discretized heat conduction equation for the enclosure surface using the finite-difference method, q_{ir} is the radiative heat flux from internal heat sources and solar radiation, and q_{ik} is radiative heat flux from surface *i* to surface *k* and can be seen as

$$q_{ik} = h_{ik,r}(T_i - T_k) \quad (3)$$

where $h_{ik,r}$ is the linearized radiative heat transfer coefficient between surfaces *i* and *k*, T_i is the temperature of internal surface *i*, and T_k is the temperature of internal surface *k*. $q_{i,c}$ is the convective heat transfer from surface *i* which can be written as

$$q_{i,c} = h_c(T_i - T_{room}) \quad (4)$$

where h_c is the convective hear transfer coefficient and T_{room} is the room air temperature.

The convective heat transfer coefficient, h_c, is unknown and estimated by empirical equations or as a constant. If the room air temperature, T_{room}, is assumed to be uniform and known, the internal surface temperatures, T_i, can be determined by simultaneously solving the energy balance equation for a surface. Space heating and cooling can then be determined from the energy balance equation for room air. It can be seen that the internal convective heat transfer from enclosures is the explicit linkage between room air and surface energy balance equations. Its accuracy will directly affect the energy calculation.

2.2 Computer fluid dynamic principles

CFD uses numerical methods to solve the Navier-Stokes equations for fluid flow and also solves the conservation equation of mass for the contaminant species and the conservative equation of energy for building thermal comfort and indoor air quality analysis. All the governing conservation equations can be seen in a simple form as following;

$$\frac{\partial \Phi}{\partial t} + (V \cdot \nabla)\Phi - \Gamma_{\phi}\nabla^2\Phi = S_{\phi} \quad (5)$$

where Φ is the V_j of the air velocity component in the *j* direction, I for mass continuity, *T* for temperature, *C* for different gas contaminants, and *t* for time, *V* is the velocity vector, Γ_{ϕ} is the diffusion coefficient, and S_{ϕ} is the source term. Φ and *C* could also stand for turbulence parameters and water vapor and various gaseous contaminants, respectively. For buoyancy-driven flows, CFD usually employs the Boussinesq approximation. The buoyancy-driven force is treated as a source term in the momentum equations. Because most practical flows are turbulent, a turbulence model must be applied for most indoor airflow to make the flow solvable.

It is impossible to obtain analytical solutions for room airflow because the governing equations are highly non-linear and self-coupled. Therefore, CFD calculates the equations by discretizing the equations with the finite-volume method. The spatial continuum is divided into a finite number of discrete cells, and finite time-steps are used for dynamic problems. The discrete equations can be solved together with the corresponding boundary conditions. Iteration is necessary to achieve a converged solution (Patankar, 1980). The accuracy of CFD calculation for indoor airflows is most sensitive to the boundary conditions supplied by users which relates to the inlet, outlet, enclosure surfaces, and internal objects. The velocity, temperature and turbulence of the air entering from diffusers or windows determine the inlet conditions, while the internal surface convective heat transfers in terms of surface temperatures or heat fluxes are for the enclosures. The accuracy of the CFD results crucially depends on these boundary conditions.

2.3 Coupling strategies

The convective heat transfer from internal surfaces is equally important to both ES and CFD. ES needs accurate convective heat transfer coefficient and room air temperature calculated by CFD and CFD requires internal surface temperatures determined by ES. Therefore, to improve their accuracy it is necessary to couple the ES and CFD application. The air temperature in the boundary condition of a surface and the convective heat transfer coefficient are two important parameters determining the convective heat transfer. However, most ES applications assume a complete mixing in room air in solving the energy balance equation. CFD can provide the air temperatures close to the surfaces from the air temperature distribution, and the convective heat transfer coefficients as

$$h_{i,c} = C_p \frac{\mu_{eff}}{Pr} \frac{1}{\Delta x} \quad (6)$$

where C_p is the specific heat capacity of air, μ_{eff} is the effective kinetic viscosity, *Pr* is the Prandtl number, and Δx is the normal distance from a point near a wall to the wall. A straightforward coupling method is to pass the air temperature, $T_{i,air}$, closed to a wall surface and the corresponding averaged convective heat transfer coefficient, $h_{i,c}$, to ES. Thus,

$$q_{i,c} = h_{i,c}\left(T_i - T_{i,air}\right) = h_{i,c}(T_i - T_{room}) - h_{i,c}\left(T_{i,air} - T_{room}\right) \quad (7)$$

where T_{room} is the desired air temperature of the room. ES use the updated $T_{i,air}$ and $h_{i,c}$ from each call of a CFD program and substitute them into the above equation. Then by solving the energy balance equation for room air and surface wall together with the above equation, the surface temperatures and heat extraction can be used to update the boundary conditions for the next CFD run. In principle, a fully iterated CFD and ES coupling approach can provide a solution that is equivalent to the conjugate heat transfer method (Patankar, 1980, Holmes, 1990, Chen, 1995) provided that the ES program subdivides surfaces sufficiently to model any important temperature variation (Moser, 1995). However, since the CFD coupled with ES program is not calculated in the transient simulation, the CFD solution at a specific time step is actually quasi-steady, consistent with the given boundary conditions for that time step. Such an approach, thus, has the advantage that it does not attempt to solve the flow filed during the transition from one time step to the next, and thus significantly reduces on computation time. Some researchers have successfully applied this strategy to integrated calculation (Srebric, 2000, Zhai, 2002, Clarke, 1995). Since the surface temperatures and heat flows vary with time in buildings, it is important to run CFD for each time-step. At each time step, iteration between CFD and ES may be needed to reach a convergence. Figure 1 gives an overview of the structure of the coupled simulation.

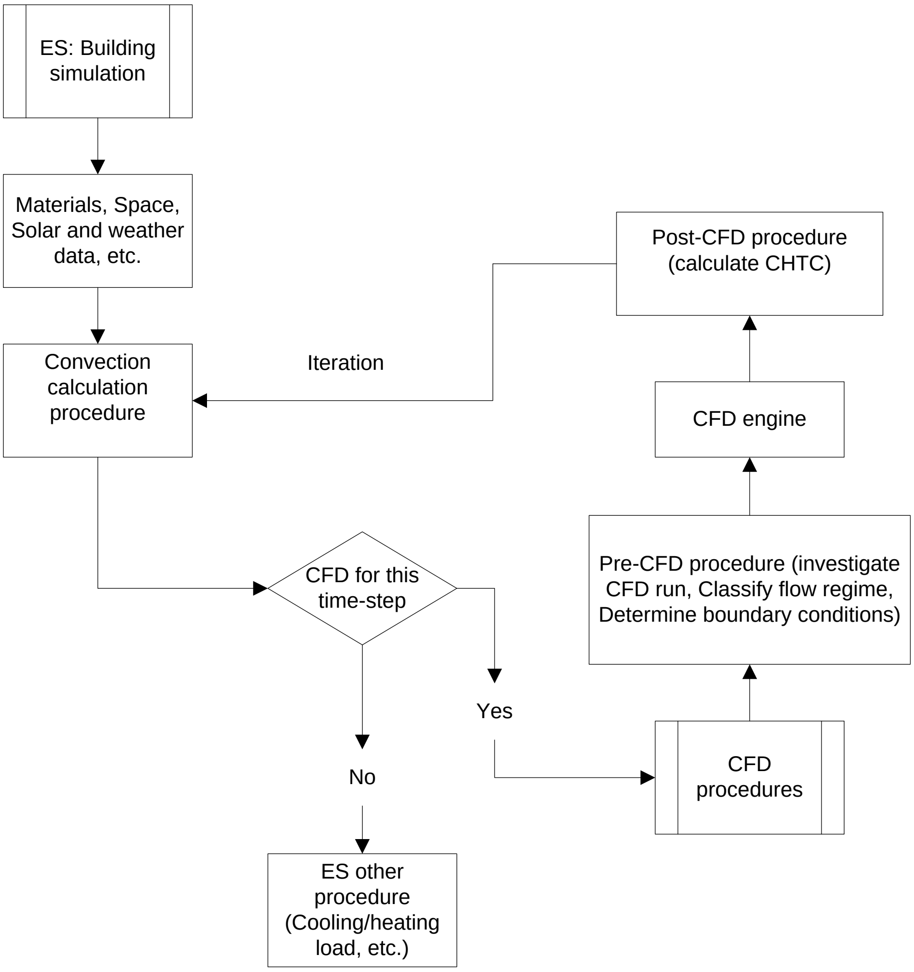

Fig.1 Coupled simulation diagram

In this study, commercial software IESVE is used as an external CFD program (MicroFlo module) to couple with ES (Apache: Thermal module). In external coupling, the two programs work independently and exchange information at the internal surface. For every time step during the calculation of CHTC of internal surfaces, the procedure checks whether there is any CFD call defined for that time step. If not, it uses another mechanism for defining the CHTC of the internal surfaces. If yes, it will invoke the coupling procedure to derive the CHTC from a CFD simulation. The coupling mechanism consists of a pre-CFD procedure, the CFD simulation engine, and a post-CFD procedure. The pre-CFD procedure is a simple CFD simulation that will classify the flow regime near each surface and decides which boundary conditions are applied for the CFD simulation. The CFD simulation mainly uses the standard k-e turbulence model. After the simulation, the post-CFD procedure will calculate the CHTC for each internal surface based on the CFD results, and decides whether the predicted CHTC can be used for further calculation. Zhai (2002) found that in the iteration between CFD and ES, the solution does exist and is unique. They reported that normally convergence can be reached after 4-10 iterations. If it takes one hour as the standard time step in most ES, it can be concluded that a 6-15 minutes time step is small enough for external surface coupling to get the same accuracy as the conjugate heat transfer method.

3. Validations of the program

To demonstrate the benefits of coupled simulations over separate applications the program has been validated with available experimental data. The case study is derived from the International Energy Agency (IEA) Annex 21/Task 12 test facility. The facility was built for providing reliable experimental data for empirical validation of building energy simulation programs. Figure 2 shows the model of the test facility built with the geometry and materials information from the IEA report. The test cell under investigation is located in Bedfordshire, UK. The site has eight semi-detached rooms, which have a lightweight, timber framed, construction. A detailed description of the site and the test room can be found in Lomas (1997). The model consists of two zones, test room and roof space. The building envelopes are exposed to the outdoor environment except the west wall that is adiabatic because of the identical adjacent room.

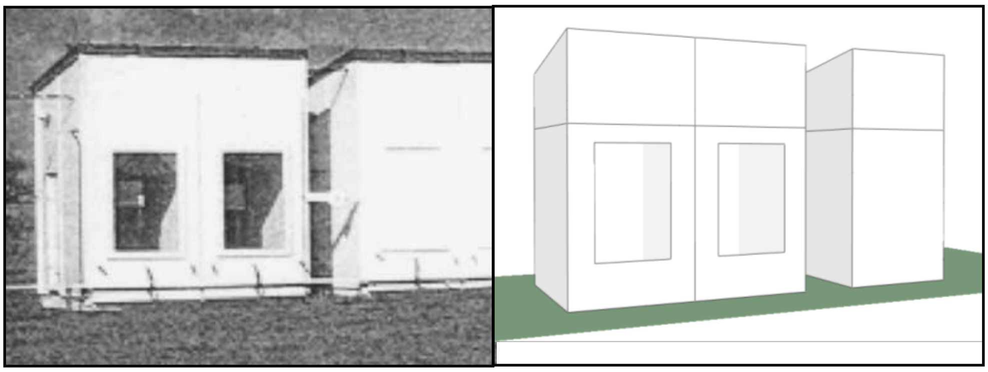

Fig.2 Test cell site (Lomas, 1997) and computer model (the middle room is the test room)

The geometry and construction is built according to the report. The following assumptions were used when creating the model; The test cells were developed in pairs. However, the wall connecting the neighboring cell is heavily insulted. Only one of the cells is modeled. The cell on the other side is explicitly modeled as it obstructs solar radiation. The ES program uses the sun tracking algorithm to calculate the solar shading. The default sky model was used to estimate the amount of diffuse solar radiation striking the building surfaces. The default insolation algorithm was used to calculate the distribution of solar beam radiation to the room's internal surfaces. The fabric thermal property data provided in the IEA report were used unmodified.

This study simulates the case of natural convection in a room without radiator with double glazing south window with a 10 min. time step, under the actual outdoor climate conditions. In the coupled simulation, grid cells were divided into 15×19×20 = 5700 cells. A dynamic coupling strategy is adopted for this study to exchange information between ES and CFD. The simulation was set at each hour for 10 days. The first 3 days were a start-up period to minimize the effect of the initial conditions. The results of the last 7 days were then analyzed.

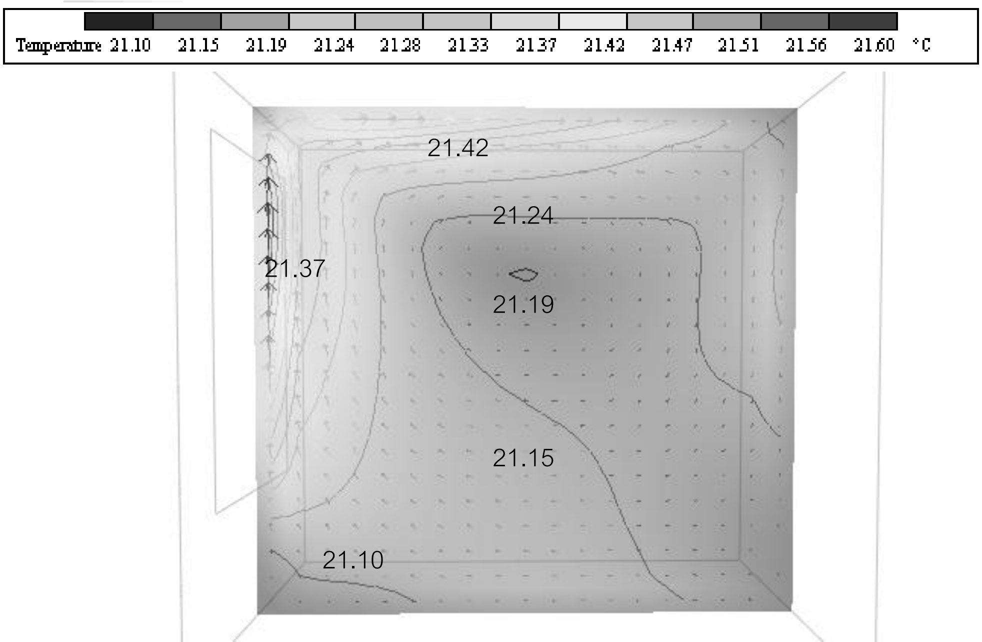

Fig.3 Temperature and airflow pattern in the middle plane of the IEA test room without radiator (10:00am)

Figure 3 shows the distribution of indoor air temperature in the middle plane of the room at 10:00 in a typical test day, computed by CFD with the boundary conditions obtained from ES. The figures show how the outdoor conditions influence the indoor airflow patterns through the window. The indoor air temperature has a very small gradient within the room. The same uniform temperature patterns were observed in the measurement, which implies that the uniform air assumption of ES may be acceptable for this case. Figure 4 compares the measured and computed air temperature at the center of the room for 1 day. The result shows that the ES can produce reasonable solutions for this case. However, the coupled results agree better with the measurement by capturing a more precise peak room temperature in the later afternoon. Negrao (1998) also reports a similar result although he used a different coupled CFD and ES simulation. The simulation of the natural convection room without a heater provides reasonable results even without coupling method because of the acceptable convection coefficient correlations used and uniform distributed indoor air temperature. Thus, a room with a heater would impose much more interesting simulation.

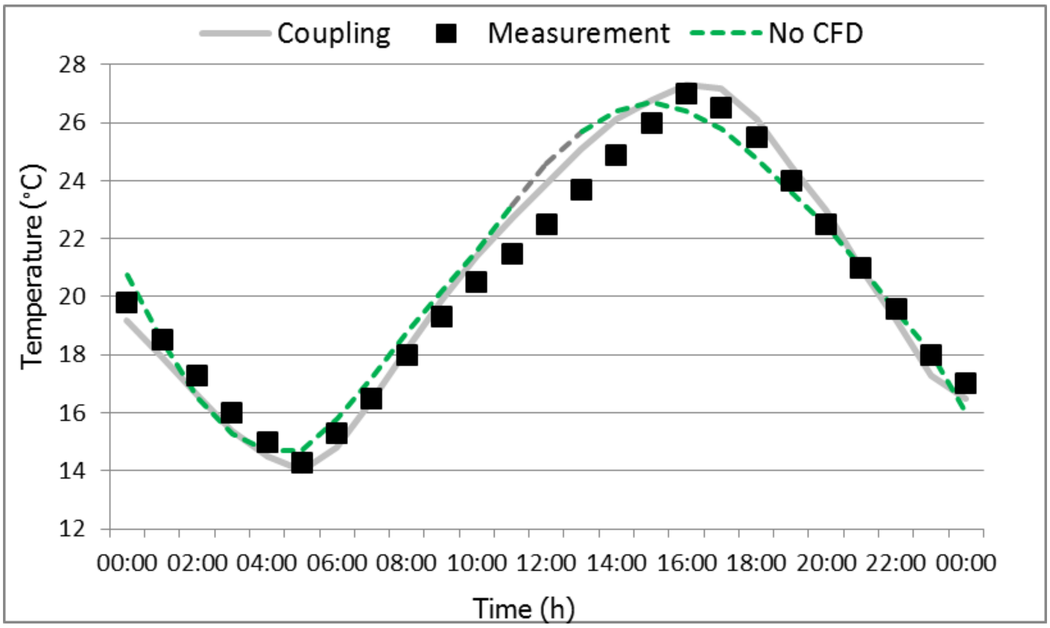

Fig.4 Measured and computed mean air temperature for the IEA test room without radiator

Lomas (1997) report some interesting results about the simulated calculation and measurement that most ES programs under-predicted the energy consumption. The prediction varied considerably between programs (52% variance in the case of the double glazed room). Furthermore, most programs under-predicted the lowest and highest temperatures in the test room. The IEA reports that the modeling of internal convection and the influence of temperature stratification are two primary causes for the discrepancies between simulated and measured results. In this study, the test room was investigated with an oil-filled electrical panel radiator located under the south window. The heat output of the radiator was 60% radiative and 40% convective with the maximum power output of 680 W. The simulation has been performed under the real climate conditions. The full dynamic coupling simulation has been run for the 10 consecutive days. Figure 5 compares the calculated and measured mean air temperatures for a single day. In the investigation, the heater turned on at 6:00 and, because the set-point temperature of 30°C, it would not heat up the air around the sensors in the room until around 11:00. This is because the portion of radiative heat output would be absorbed by the internal surfaces before heat is convected into the air. This results in time lags in the response of the air temperature to heat injection. As can be seen from the figure, the temperature predictions are very close. Both coupling and non coupling simulations have followed the same heat transfer behaviors, although smaller time lags were found. The difference occurred when the heater is on (at 6:00) and when the heater is off (18:00). The temperature reached the steady state condition within 3-4 hours in the simulation, while in the experiment it took around 5 hours. The small time lag is probably caused by the current ES program that cannot properly model the dynamic behaviors of the oil-filled radiator. It results in the fast temperature rise and drop in the simulation. However, the air temperature predicted from the coupled simulation is closer to the measured data than that of the non coupling simulation.

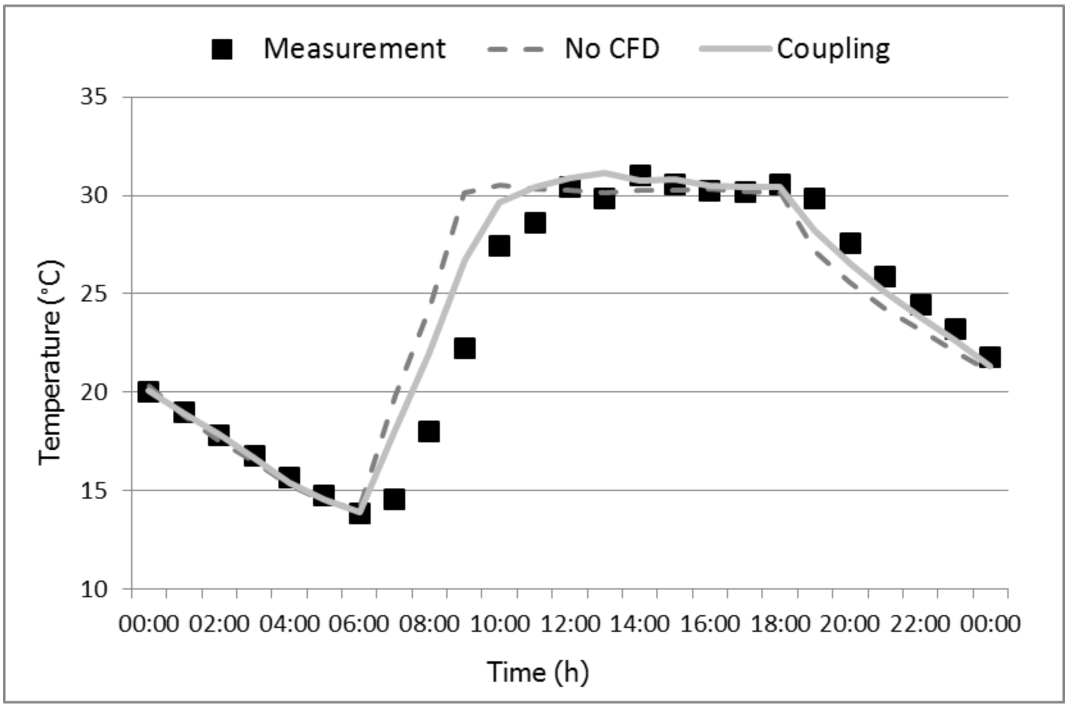

Fig.5 Measured and computed mean air temperature for the IEA test room with radiator

Figure 6 shows the energy consumptions for the whole period of simulation. The non coupled simulation results 67.7 MJ. The result from the coupled simulations are much closer to the measurement than that of the non coupled simulations being 74.5 MJ comparing to the measured value of 89.4 MJ. The coupled result has slight improvement comparing to the non coupled approach which is attributed to the higher CHTC during the time when the heater is on. Although distinct CHTC and air temperature stratification were obtained from the coupled simulation, the energy consumptions of the radiator provided by the coupling approach were still under estimated. Olsen (2002) used EnergyPlus for the coupling simulation and obtained similar results. He found that this is due to the lack of any time lag in the radiator model, which allows the air to heat up faster than it actually does, which thus allows the radiator energy consumption to decrease more rapidly.

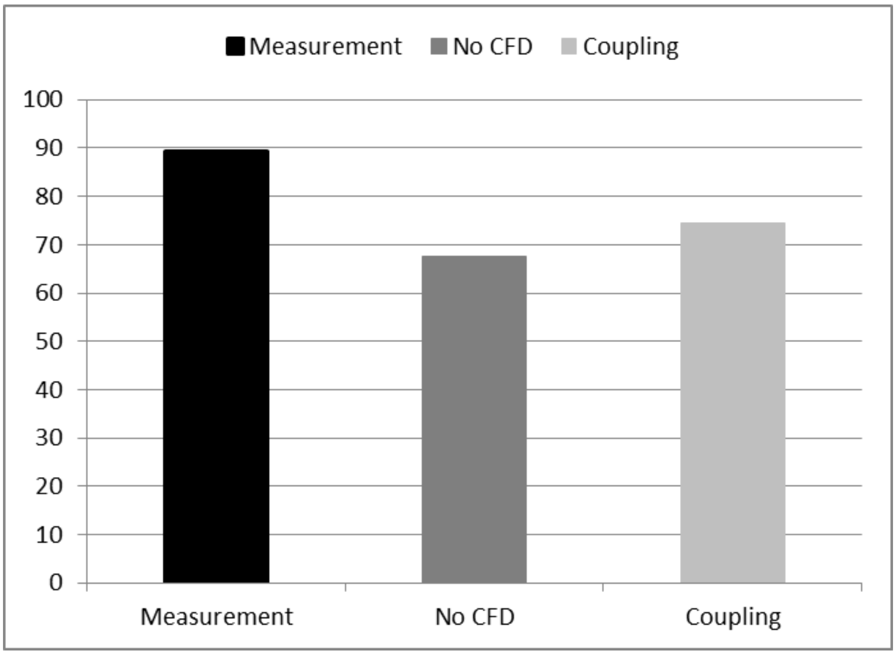

Fig.6 Energy consumption for the IEA test room with radiator

The study further validated the coupling simulation program with field measurements using the main hall building of Shinawatra University located in the rural area of Pathumthani province, Thailand. The main hall is a 5-storey building of circular shape with a central atrium and a dome skylight (see Figure 7). The building has a total floor area of 21,810 m^2 with the height and diameter of 43 m and 73 m, approximately. Its building envelop consists of EIFS (external insulation and finished system) and heat-stop glass. This double-glazed window has a low shading coefficient, low thermal conductivity and low moisture penetration. The building has been designed with an understanding of the hot-humid climate in Thailand with an attempt to minimize the energy gain, moisture penetration and air infiltration. Further details can be found in Boonyathikarn (2002). Modeling of this facility is quite challenging because the indoor environment is not well mixed due to the stratification effect of the atrium. The experimental study was only for natural ventilation in summer design day.

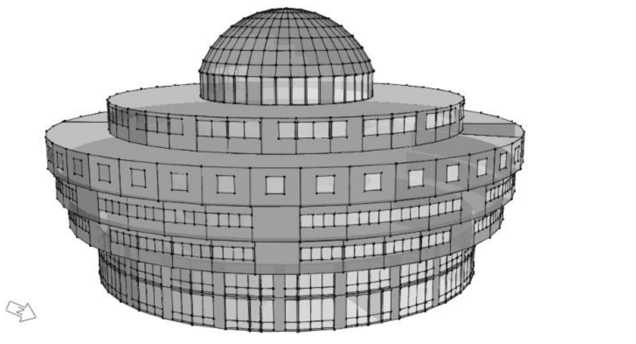

Fig.7 Main hall building of Shinawatra University

Measurement data has been collected and widely used to validate the coupling program. The measured data are such as air temperatures and surface temperatures of walls and windows for both indoor and outdoor. The data loggers and sensors have been installed in the eight orientations of the envelope (north, northeast, east, southeast, south, southwest, west and northwest directions). The indoor air temperatures of the atrium are also collected. Furthermore, hourly average meteorological data such as temperature, humidity, wind speed and direction and solar radiation can be obtained from a weather station of the University. The case has plenty of measurement data for both building envelope and indoor air, which have been widely used as a reference. To accurately measure the air and surface temperatures, the silicon temperature sensors with a measurement range of -40 to 125ºC with an error of ±0.1ºC at 0 to 100ºC are used. The resolution of the sensor is 0.1 ºC. Before collecting the data, the main hall and its operation is shut down for 48h in order to stabilize the building to a passive state after normal operation. The data were collected every 10 min. during the semester break (March-May and October, 2007), all collected data can be used, while only data collected during the weekends are used during the regular semester. In the coupled simulation, the model of CFD has been created with both uniform and non-uniform grids with a total number of cells of 8,037,360 (244 x 244 x 135).

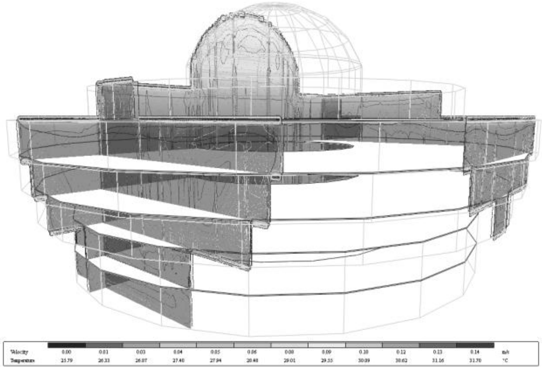

Fig.8 Temperature and velocity profile of Shinawatra University's atrium

Figure 8 illustrates an example of CFD coupling results for temperature and velocity profile of the atrium at various sections. The measured and simulated indoor air temperatures at the center of the atrium are showed in Figure 9. The trends of the indoor air temperatures calculated from both coupled and non coupled simulation varied during the day and agreed with the measurement. The peak indoor air temperature was found between 17:00-18:00. The time lag responses and the peak delays are due to the thermal capacity and thermal mass of the EIFS wall. The coupled simulation resulted a higher peak temperature than the non coupled simulation by approximately 0.7 ºC. This is mainly because the non coupled simulation underestimates the convective transfer coefficient from the enclosures. Table 1 shows the convective heat transfer coefficient from the windows in the eight orientations. A big difference of the convective heat transfer coefficient is found, where the difference is about 4-6 times. Consequently, the surface temperature of the interior window from the coupled simulation is closer to the measurement than that of the non coupled simulation (Figure 10). The distinction of window temperatures between two simulations is not as much as that of the convective heat, implying that in this case the radiation between the surfaces contributes significantly. In general, the coupled simulation results show a better agreement with the measurement.

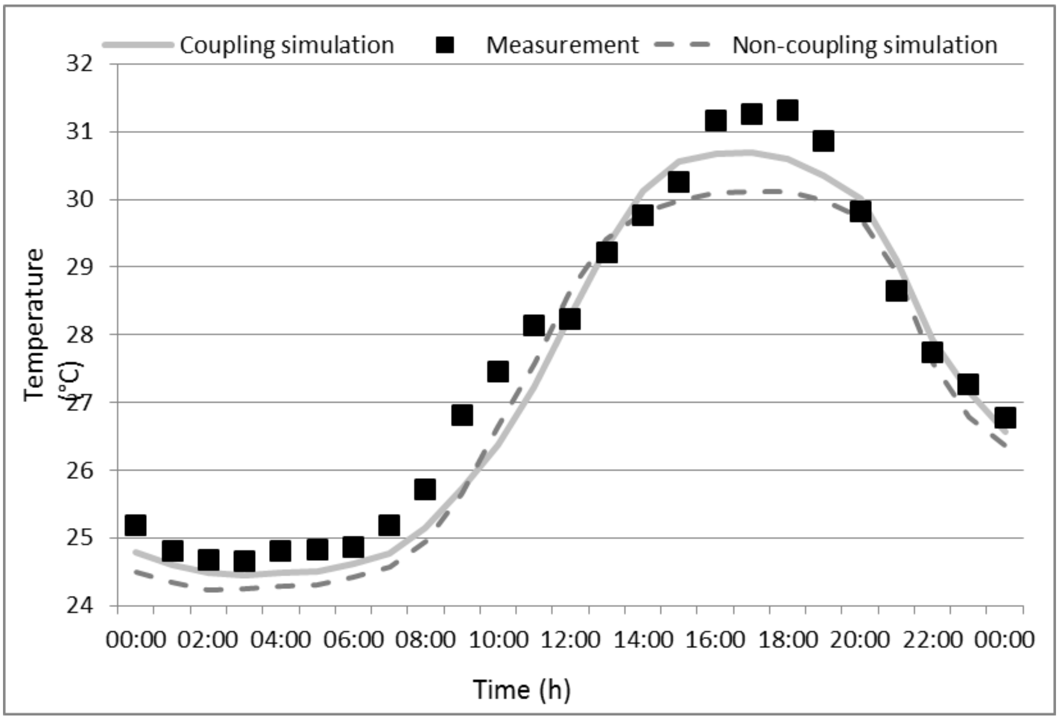

Fig.9 Measured and computed temperature profile of the atrium

Enclosure surfaces	Convective heat transfer coefficient (W/m^2C) CFD coupling	Non coupling
North	4.17	1.44
Northest	11.28	2.32
East	16.48	2.86
Southeast	14.62	2.54
South	8.32	2.16
Southwest	14.89	2.51
West	16.03	2.74
Northwest	9.24	2.23

Table 1. Comparison of convective heat transfer coefficient of windows with and without coupling simulation of Shinawatra University

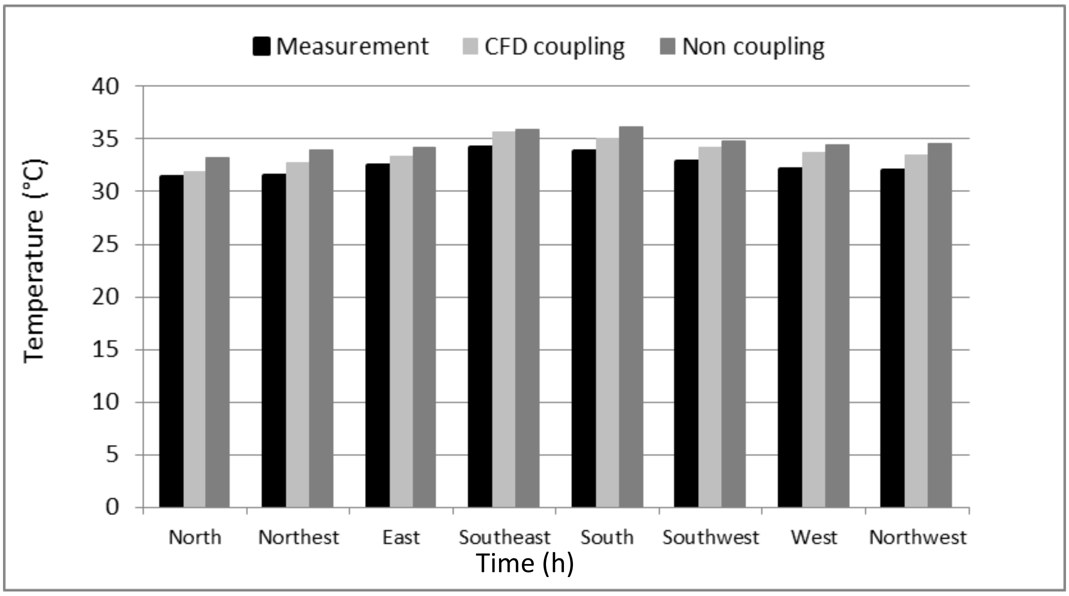

Fig.10 Measurement and simulation results of interior surface temperature of windows

Conclusions

This study develops and outlines the strategy to externally couple CFD computation and building energy simulation. With the coupling approach, most assumptions used by the two programs for flow boundary and thermal conditions can be eliminated due to the complementary information provided by the two applications. The investigation has validated the coupled program with the experimental data obtained from the literature. The results verify that the simulation approach can provided reasonable and reliable calculation on building thermal predictions. In general, the coupled simulation gives more accurate and detailed results than the separate simulations. Mainly, this is because ES obtains more accurate convective heat from boundary conditions and can provide more accurate calculation of thermal behaviors of building envelopes. Thus, CFD program can receive more precise and real time thermal boundary conditions and can predict the dynamic indoor environmental conditions that are important for the assessment of indoor air quality and thermal comfort. The study further validates the coupled simulation with experimental data of environmental indoor conditions from the atrium of Shinawatra University, Thailand. The results reveal that indoor air temperature gradient and convective heat transfer coefficient have great impact on the building simulation. The empirical coefficient correlations used by ordinary ES software may significantly deviate from the real values. The coupled approach should be used for buildings with large indoor air temperature stratification.

Acknowledgements

The Thailand Research Fund (TRF) and Shinawatra University are gratefully acknowledged for funding this research.

References

1. Negrao, C.O.R. (1998) Integration of computational fluid dynamics with building thermal and mass flow simulation. *Energy and buildings,* 27, 155-165.

2. Srebric, J. Chen, Q. and Glicksman, L.G. (2000) A coupled airflow-and-energy simulation program for indoor thermal environmental studies. *ASHRAE transactions,* 106(1), 465-476.

3. Beausoleil, I.M. (2002) The adaptive conflation of computational fluid dynamics with whole-building thermal simulation. *Energy and buildings,* 34, 857-871.

4. Zhai, Z. and Qingyan, C. (2003) Solution characters of iterative coupling between energy simulation and CFD programs. *Energy and buildings,* 35, 493-505.

5. Tan, G. and Leon, R. (2005) Application of integrating multi-zone model with CFD simulation to natural ventilation prediction. *Energy and buildings,* 37, 1049-1057.

6. Cheong, K.W.D. Djunaedy, E. Poh, T.K. Tham, K.W. et al. (2003) Measurements and computations of contaminant's distribution in an office environment. *Building and environment,* 38(1), 135-145.

7. Patankar, S.V. (1980) *Numerical heat transfer and fluid flow*, McGraw-Hill, New York, Hemisphere.

8. Holmes, M.J. Lam, JK.W. Ruddick, K.G. and Whittle, G.E. (1990) Computation of conduction, convection, and radiation in the perimeter zone of an office space. *Proceedings of ROOMVENT '90*, Norway.

9 .Chen, Q. Peng, X. and Van Paassen, A.H.C. (1995) Prediction of room thermal response by CFD technique with conjugate heat transfer and radiation models. *ASHRAE transactions,* 3884, 50–60.

10. Moser, A. Schalin, A. OB, F. and Yuan, X. (1995) Numerical modeling of heat transfer by radiation and convection in an atrium with thermal inertia. *ASHRAE transactions,* 101(2), Meeting SD-95-14-4, 1136–1143.

11. Zhai, Z. Chen Haves, P. and Klems, J.H. (2002) On approaches to couple energy simulation and computational fluid dynamics programs. *Building* and environment, 35, 857-864.

12. Clarke, J.A. Dempster, W.M. and Negrao, C.O.R. (1995) The implementation of a computational fluid dynamics algorithm within the ESP-r system. *Proceedings of building simulation '95*, 166–175.

13. Chen, Q. and Xu, W. (1998) A zero-equation turbulence model for indoor airflow simulation. *Energy and Buildings,* 28, 137–44.

14. Lomas, K.J. Eppel, H. Martin, C.J. and Bloomfield, D.P. (1997) Empirical validation of building energy simulation programs. *Energy and buildings,* 26, 253-275.

15. Olsen, E.L. (2002) Performance comparison of UK low-energy cooling systems by energy simulation. *Master thesis, Massachusetts Institute of Technology*, USA.

16. Boonyathikarn, S. (2002) *Shinawatra University: an integrated design*, GM Max Media, Bangkok.

List of figures

เอกสารแนบหมายเลข 3

Output จากโครงการวิจัยที่ได้รับทุนจาก สกว.

1. ผลงานตีพิมพ์ในวารสารวิชาการนานาชาติ (รอการตอบรับจาก Building Research and Information Journal)
2. การนำผลงานวิจัยไปใช้ประโยชน์
   - เชิงสาธารณะ
     ได้รับความสนใจในวงกว้างจากกลุ่มบริษัทอสังหาริมทรัพย์ ที่จะได้ประโยชน์จากการนำ Coupled building energy and CFD simulation ไปใช้ในการวิเคราะห์ พัฒนา และออกแบบ อาคารและบ้านให้ใช้พลังงานลดลง
   - เชิงวิชาการ
     มีการพัฒนาการเรียนการสอนโดยใช้ Coupled building energy and CFD simulation