COUPLED ENERGYPLUS AND COMPUTATIONAL FLUID DYNAMICS NATURAL VENTILATION SIMULATION

Rui Zhang¹, Khee Poh Lam², Shi-Chune Yao², Yongjie Zhang²
¹IBM, T.J. Watson Research Center, Yorktown Heights, NY, 10598
²Carnegie Mellon University, Pittsburgh, PA 15213, USA

ABSTRACT
Energy modeling approaches have continued to advance to cater for emerging new design concepts towards “greener” solutions that optimize energy consumption in buildings while maintaining thermal comfort as well as healthy environment. Increasing attention is given to passive and mix-mode systems in building. Computational Fluid Dynamics (CFD) model has been widely adopted as effective tool for natural ventilation simulations. However, CFD become unstable for conjugate heat transfer model, which is the transient heat transfer between solid, e.g., walls, and fluid, e.g., air. Solid and fluid has different respond times. Typically walls respond in hours, and air responds in seconds, causing the system to become stiff. In addressing the issue, a coupled lumped heat transfer model (EnergyPlus) and CFD model (Fluent) was implemented, and 8 days of simulation was conducted. The airflow rates of openings from the airflow network module in EnergyPlus and airflow rates from CFD model were compared. Results show that airflow network model generally predict smaller airflow rates for the openings. Airflow network model generates better results for openings on the south and east facade and internal openings, with difference ranging from −100% to −200%.

INTRODUCTION
Energy modeling approaches have continued to advance to cater for emerging new design concepts towards “greener” solutions that optimize energy consumption in buildings while maintaining thermal comfort as well as healthy environment. Instead of the conventional approaches that rely solely on mechanical system to provide the desired thermal conditions, increasing attention is given to passive and mix-mode systems in building. Nevertheless, advances in system modeling, there are still limitations in the capability of representing the “real” environmental behavior with different building spatial configurations. For example, most current energy models adopt a “nodal” approach to simulate the heat transfer process in the building, with simplified heat resistors and capacitors network. The underlying assumption for nodal model is the uniformity of room temperature, which may apply to most of the building zones with moderate room size. However, limitations also come with such nodal approach:

- Priori and empirical knowledge of various coefficients are needed for model input, such as wind pressure coefficient, heat transfer coefficient, loss factors, friction factors, etc.
- Effects of thermal and air flow patterns introduced by building spatial configurations are difficult to be represented into the model.

More detailed zonal model, such as COMIS ((Feustel and Rayner-Hooson 1990)), has been developed for ventilation design in complex buildings. However, research (Mora, Gadgil, and Wurtz (2003)) shows that the results of zonal model are not satisfactory compared with even coarse-grid CFD models under isothermal conditions.

Finite Volume Method (FVM) such as the Computational Fluid Dynamics (CFD) on the contrary will perform detailed computation on the heat transfer and air flow simulation, which could supplement the nodal model for the building energy simulation. Detailed temperature profile and air flow field are calculated with first principle based Navier-Stokes set of equations and turbulence models. Coefficients, such as the heat transfer coefficients, will be the results of the simulation, which is defined by a set of boundary conditions. However, limitation also comes with CFD. Firstly, the computation resources needed for CFD simulation are much more expensive than the nodal model. Secondly, the coupling between fluid and solid material is difficult and computational more expensive, thus the thermal storage capacity of building component is difficult to be correctly modeled. Therefore, CFD has been limited to detailed studies of given space under a number of specific boundary conditions. However, with the rapid development of computation power, the computation time of CFD is decreasing, it is highly necessary to develop an integrated thermal simulation that combines the advantages of both the nodal and FVM model, providing higher accuracy within acceptable computation time.
The nodal model calculation of the heat transfer process in the building is based on the fundamental heat balance principles. Each zone is assumed with uniform state variables of temperature, pressure and density. Empirical coefficients are used to calculate the state variable transfer fluxes between each node. One example of how such nodal system is implemented is shown as follows:

\[
C_i \frac{dT_i}{dt} = \sum_{i=1}^{N_{di}} \dot{Q}_i + \sum_{i=1}^{N_{faces}} h_{Ai}(T_{ai} - T_{i}) + \sum_{i=1}^{N_{max}} m_{ci}C_p(T_{ci} - T_{i}) + m_{sys}C_p(T_{sys} - T_{i}) + \dot{Q}_{sys}
\]

Where:

- \(C_i \frac{dT_i}{dt}\): room air energy change rate
- \(\sum_{i=1}^{N_{di}} \dot{Q}_i\): sum of the convective heat transferred through internal heat sources or sinks,
- \(\sum_{i=1}^{N_{faces}} h_{Ai}(T_{ai} - T_{i})\): sum of the convective heat transferred through building envelope,
- \(\sum_{i=1}^{N_{max}} m_{ci}C_p(T_{ci} - T_{i})\): total energy of infiltration air,
- \(m_{sys}C_p(T_{sys} - T_{i})\): energy from neighbor zones air mixing system output,
- \(\dot{Q}_{sys}\): energy from neighbor zones air mixing system output.

In equation 1, the heat transfer coefficient needs to be determined for a given set of boundary conditions that define the thermal and fluid fields in the space. Various empirical correlations have been developed to calculate the heat transfer coefficient, which are dependent on the ambient air flow characteristics. However, the lack of knowledge of the air flow, will lead to considerable errors in the heat transfer prediction of the nodal model for buildings, where the heat transfer process is complex by nature. Research (Zhai et al. 2002)) shows that differences in heat transfer coefficient between CFD and empirical equation can range from 1.42 W/m²K to 111.41 W/m²K. Mora, Gadgil, and Wurtz (2003) investigated the effect of heat transfer coefficient between CFD and empirical equation. The results of the air temperature in the void space and the surface temperatures of the corridors were compared with measured data. Results generated by the nodal were generally lower than the measured data.

**PREVIOUS WORKS**

Manual coupling of the CFD model and nodal model has been done for many geometrical complicated buildings. Nagai and Kurabuchi (2009) used CFD to decide the coefficients in the nodal model for a high-apartment building with central void space throughout the height of the building in Japan. Manz and Frank (2005) used an one-way-static coupling to study the thermal performance of double facade buildings. Wong et al. (2005) also coupled CFD and nodal model to study the performance of double facade building in a tropical climate in Singapore.

Research work on automated coupling of the nodal model and CFD model at run time is dated from 1990s. Negrao (1995) implemented a full iterative coupling approach of the nodal model and CFD model. Selected space was simulated with CFD, and ESP-r was used as the nodal model. The coupled variables were: surface temperature of all the walls and windows, and the pressures at the internal openings that connect the space with the rest of the building. A full iterative strategy has been used, where coupled variables will be exchanged at each iterative step until a convergence criterion was reached at each time step. It was found that low under-relaxation (0.1) was necessary in order for the CFD computation to converge for the coupled system. The number of iteration of the CFD computation was increased compared with the uncoupled CFD computation.

Based on the work by Negrao (1995, Negrao 1998), Beausoleil-Morrison (2000, Beausoleil-Morrison et al. 2001, Beausoleil-Morrison and Clarke (1998) continued with the investigation of the coupling between nodal model (ESP-r) and CFD model. A conflation controller was implemented to configure the CFD model at each time step. At the start of each time step, the zero-equation turbulence model was employed, and the resulting viscosity field was used to initialize the \(k - \varepsilon\) turbulence model.
model. Grashof Number (the ratio of buoyancy and viscous forces) and Reynolds Number (the ratio of inertial and viscous forces) were evaluated. The existence of the buoyancy term in z-momentum equation was determined. The conflation controller would then choose the appropriate turbulence model and near wall model. An empirical validation of the coupled model by Beausoleil-Morrison (2000) is conducted by Bartak et al. (2002). Two CFD models with coarse-grid and fine-grid were constructed and compared with experimental results. It was found that the coarse-grid and fine-grid CFD produced comparable results. Variables validated were air temperature, comfort parameters, and local mean age of air under isothermal and non-isothermal conditions. Good agreements were found between the predicted and measured data.

Djunaedy, Hensen, and Loomans (2003), Djunaedy, Hensen, and Loomans (2004b), Chen, Peng, and van Passen (1995) and Zhai et al. (2001) discussed both the advantages and disadvantages of internal coupling of the CFD and nodal model. It was concluded that external coupling is more favorable in terms of no stiffness issue, less expensive, nodal and CFD model themselves are independent and can be optimized individually.

Zhai and Chen (Zhai et al. 2002, Zhai and Chen 2003), Zhai, Gao, and Chen (2004), Zhai (2004), Zhai (2005), Zhai (2006) also investigate the coupling strategy extensively. They implemented a coupling strategy to exchange heat transfer coefficient and surface boundary conditions between nodal model and CFD model through dynamic or static coupling. Hereby, static and dynamic is the strategy used for data exchange instead of thermal process. A sensitivity study was also carried out to investigate the effectiveness of the coupling method in improving the building energy simulations. Their results showed that for rooms with moderated size, without sensible temperature stratification, the coupling approach shows marginal effect. However for rooms with large temperature stratification, the discrepancy between coupled method and nodal model is significant (42%). Wang (2007), Wang and Chen (2005) proved theoretically that the coupled model has a solution and it is unique. The investigation used Scarborough criterion to evaluate convergence performances and analyzed the stabilities of three coupling strategies, which are pressure coupling, pressure and flow rate mixed coupling, and flow rate coupling. It was found that the pressure coupling performs best.

Wang and Wong (2008) developed a text-based interface for automated coupled program to extract and exchange information between TAS (nodal model) and Fluent (CFD model). External coupling was used and results showed that coupled program yield better results compared with single nodal or CFD model. Wang and Wong (2009) compared velocity inlet coupled and pressure inlet coupled strategy, and found that pressure inlet coupled strategy gave better results.

**COUPLING METHOD**

EnergyPlus is chosen as the nodal model for the coupled simulation platform. EnergyPlus is a program to calculate the energy required for heating and cooling a building using a variety of systems and energy sources over a year’s period. The Fluent software is chosen as the FVM model for the coupled simulation platform. Fluent uses FVM method, which is usually referred to as Computational Fluid Dynamics (CFD), to solve the fluid dynamic problems. Fluent software is one of the most widely used and extensively tested software in the architectural applications domain. It provides multiple modelling capabilities of flow, turbulence, heat transfer, thermal radiation, solar module, and mass transfer, etc.

**Coupling Strategy**

The coupling approaches between the nodal and FVM model can be characterized into three types:

- Full internal coupling, where the set of equations for nodal model and CFD model are solved together iteratively. Research (Negrao (1998)) shows that such internal coupling will generate a cluster of equations referring to different sub-systems (such as building zones, CFD equations and plant systems), which is large and sparse. If established, this large sparse matrix would require the square of the sum of the sub-system matrices for its storage.
- Iterative external coupling, where the set of equations for nodal model and set of equations for CFD model are solved individually, the variables are exchanged with an iterative procedure until a converged state is achieved.
- Progressive-Replacement external coupling, where the set of variables are exchanged after each model comes to a converged state at each time step.

Research (Negrao (1998)) shows that the fully internal coupling and iterative external coupling will require larger number of iterations to reach convergence, e.g., approximately 600 iterations were necessary for a simulation which required approximately 200 iterations for an individual CFD simulation. It is also found that the convergence of the iterative approach is difficult even for simple room simulation. One of the conclusions from the above study is that the nodal model and CFD model can be satisfactorily achieved by maintaining each method’s solution algorithm separately. Furthermore, research (Djunaedy, Hensen, and Loomans 2004a) found that difference in simulation results between internal coupling and external coupling is not significant. However, the benefits of the external coupling are three folds:
  - Computationally less expensive.
• Nodal and CFD model can be maintained and updated individually.
• Configurable exchange variables.

This study will implement the Progressive-Replacement external coupling strategy. The exchange variables will be exchanged after each model, i.e., the nodal and the CFD model, comes to its converged state at each time step.

### Coupling Variables

As previously mentioned, nodal model requires various building specific heat and mass transfer coefficients, which can be obtained from CFD simulation. At the same time, the CFD tool may have stiffness problem in solving transient conjugate heat transfer with conduction and convection between the building envelope components and air in the space. Based on the characteristics of the two models, a collection of coupling variables was selected.

The nodal model will provide to the CFD tool all the interior and exterior surface temperatures of the envelope components of the building and outdoor weather conditions. The CFD tool will take temperatures of various building surfaces as settings of the thermal boundary conditions, including both interior surfaces and exterior surfaces. For natural ventilation, wind is the driving force of the flow through the openings in the building. The wind conditions, in terms of speed and direction, provided by the nodal tool from weather data will be used to determine the boundary type of the four boundary surfaces of the simulation domain. For example, if the wind is coming from south, with a speed of 2 m/s, the surface of the south bound will be set as velocity inlet, with incoming wind velocity of 2 m/s. The north bounding surface of the domain will be set as outflow. The east and west boundaries will be set as symmetry assuming there is no shear strain on the surface.

The CFD tool will conduct the steady state natural ventilation simulation. A post processing program will be implemented to extract the temperature profile and velocity fields from CFD tool and then to calculate the values for the exchange variables, which include the following:

- Airflow rates through all the openings.
- Average air temperatures through the openings.
- Surface heat transfer coefficients for all the envelope surfaces.

### Coupling Platform

The coupling platform will be based on Building Controls Virtual Test Bed (BCVTB (Wetter et al. 2011)), which is a software environment targeted to provide an integration platform for various simulation tools. BCVTB allows expert users to couple different simulation programs for distributed simulation or for a real-time simulation that is connected to a building control system. For example, the BCVTB enables concurrent energy simulation of whole building in EnergyPlus with HVAC system and operating control in c++ program, or MATLAB/Simulink, while exchanging data between the programs at each time step. The BCVTB is based on the open-source Ptolemy II software environment from University of California at Berkeley (UCB). The BCVTB is still under development and aimed at expert users of simulation.

### EnergyPlus object in support of the coupling

In this study a new object called the "ExternalInterface:Airflow" will be added for the purpose of coupling between nodal and CFD model for air flow simulations. The entry of the objects are designed as follows:

```plaintext
ExternalInterface:Airflow,
  <name object to set the airflow rate of the opening from externalInterface>
  "min-fields 3"
  A1, "field Opening Surface Name"
  "required-field"
  \type alpha
  \note this name is the opening surface name
  A2, "field Zone name that the air is flowing in through the opening"
  "required-field"
  \type alpha
  \note airflow to the zone after the warm-up and system sizing.
  N1, "field Optional Initial value"
  \type real
  \note If specified, it is used during warm-up and system sizing.
  \note If not specified, then the airflow only writes the airflow to the zone after the warm-up and system sizing.
```

The `ExternalInterface:Airflow` object is designed to set the airflow rate for each piece of the openings in the building. The input entry of `ExternalInterface:Airflow` will specify the name of the opening, and zone name that the air is flowing in (positive value) or flowing out from (negative value). The value of heat transfer coefficient will be updated through the `ExternalInterface:Schedule` object.

### FlowPlus program for executing CFD simulation and extracting coupling variables

A c++ program, called `FlowPlus`, was developed to execute the Fluent software to conduct the CFD simulation and extract the results for the coupling variables. The program will read in the variable configuration file, which specifies the exchange variables that EnergyPlus is sending and also the variables that are needed to be extracted from Fluent simulation results. Then the program will use the values obtained from EnergyPlus to generate a Fluent journal file, which will set the boundary conditions for the CFD simulation. Then Fluent software will be called to execute the CFD simulation according to the boundary conditions that are specified in the journal file. After Fluent finished executing the iteration, the `FlowPlus` will extract the temperature profile and velocity fields from Fluent and calculate the values for the exchange variables and send the values to EnergyPlus through BCVTB.
The variable configuration file

The variable configuration file is the key component used to specify the exchange variables. The file follows XML file format. A XML schema for the variable configuration file was defined as follows:

```xml
<?xml version="1.0" encoding="UTF-8"?>
<!DOCTYPE BCVTB-variables SYSTEM "variables.dtd">
<!ELEMENT BCVTB-variables (variable*)>
<!ELEMENT variable ( EnergyPlus* ) >
<!ATTLIST EnergyPlus name CDATA #IMPLIED>
<!ATTLIST EnergyPlus airflowtemp CDATA #IMPLIED>
<!ATTLIST EnergyPlus airflowrate CDATA #IMPLIED>
<!ATTLIST EnergyPlus variable CDATA #IMPLIED>
<!ATTLIST EnergyPlus actuator CDATA #IMPLIED>
<!ATTLIST EnergyPlus schedule CDATA #IMPLIED>
</variable>
</BCVTB-variables>
```

The attribute name and type together provide the name and type of the variables that EnergyPlus will export. The schedule, actuator and variable correspond to ExternalInterface : Schedule, ExternalInterface : Actuator and ExternalInterface : Variable respectively. The airflowrate and airflowtemp together will be used by the ExternalInterface : Airflow object in EnergyPlus to update the flow rates and flow temperature. The FlowPlus program will report the airflow rates and airflow temperature for the two elements respectively. The sequence of the exchange variable is organized as the same as the sequence of elements in the variable configuration file. Then EnergyPlus and FlowPlus will assemble and disassemble the exchanged data according to the variable configuration file. An example of the variable configuration file is as follows:

```xml
<?xml version="1.0" encoding="ISO-8859-1"?>
<BCVTB-variables>
  <variable source="EnergyPlus">
    <EnergyPlus name="ENVIRONMENT" type="OUTDOOR DRY BULB"/>
  </variable>
  <variable source="EnergyPlus">
    <EnergyPlus name="ENVIRONMENT" type="WIND SPEED"/>
  </variable>
  <variable source="EnergyPlus">
    <EnergyPlus name="int_Blg_661_z1_w" type="Surface Inside Temperature"/>
  </variable>
  <variable source="Ptolemy">
    <Float airflowRate="int_Blg_661_z1_win1"/>
  </variable>
  <variable source="Ptolemy">
    <Float schedule="int_Blg_661_z1_w"/>
  </variable>
</BCVTB-variables>
```

In the above example variable configuration file, EnergyPlus will export the outdoor dry bulb temperature, the wind speed, and the inside surface temperature of surface int_Blg_661_z1_w. The FlowPlus program will take the exported value from EnergyPlus and report back the airflow rate for opening int_Blg_661_z2_cc1_win1 and opening int_Blg_661_z2_cc2_win2 and the heat transfer coefficient for surface int_Blg_661_z1_w.

Coupling Process

The BCVTB program will invoke the coupled simulation by calling both the EnergyPlus and FlowPlus. First, EnergyPlus will start the initialization process and conduct the simulation for the first time step. After EnergyPlus finishes computation for the first time step, EnergyPlus will send the variable values through BCVTB to FlowPlus and halt till it receives data back from BCVTB for next time step. On receiving the data FlowPlus will start the steady state CFD simulation and wait till a converged stage of the steady state CFD simulation is reached. The FlowPlus program will then extract the values for all the defined exchange variables, i.e., flow rates of openings, average temperature of the flow and surface heat transfer coefficients, and send the data to EnergyPlus through BCVTB. The above process finishes one cycle of data exchange. To continue, on receiving the data back from BCVTB, EnergyPlus will compute the next time step for the next cycle of data exchange and so on.

CASE STUDY

The live retrofit building 661 on the Philadelphia Navy Yard was chosen for the case study of the coupling platform and methodology developed in the previous section. This historic building features a shared open space at the back with gross area of about 1600m², and a two story space in the front with gross area of about 800m² for each floor. After retrofit, the building will house GPIC personnel, and function as living laboratories for developing tools and methods to transform the building industry’s current fragmented serial method into integrated team efforts. The building model is shown in Fig. 1.

Simulation Scenarios Selection

The simulation of natural ventilation was chosen as the case study on the coupling between the nodal model and the CFD model. Natural ventilation is considered to be most complicated in terms of airflow simulations, and it is a very important strategy for energy efficient buildings. The accurate simulation of natural ventilation will provide architects and engineers with more insight during the design process of the energy efficient buildings.

The weather condition in Philadelphia is first analyzed. Hours when outdoor weather conditions satisfy the criteria of ASHRAE Fundamentals Comfort Model were identified. The month of June has the highest number of hours that outdoor conditions are comfortable, and potentially natural ventilation can be deployed. Thus the month of June is chosen as the simulation period for the study of coupling between nodal model and the coupled model.

RESULTS

The airflow rates calculation method by the nodal model in EnergyPlus, and the airflow rates calculation
method by the coupled CFD model were described in the following sections.

Airflow Rate Computation Method in Nodal Model

The AirflowNetwork model in EnergyPlus consists of a set of nodes connected by airflow components through linkages. Each zone is a node in the system, and openings will be linkages between the nodes. The pressure and airflow is the relationship between the components (nodes, i.e., the rooms) of the system. The pressure and airflow calculations determine pressure at each node and airflow through each linkage given wind pressure and forced airflows, (EnergyPlus 2011).

Newton’s method is used to solve for node air pressures. Starting with an initial set of values for the node pressures, an iterative process will be carried out till the mass balance of each node is approaching to zero (less than a convergence value).

Airflow Rate Computation Method in CFD Model

The CFD software Fluent deploys turbulence model with Navier-Stocks equations to solve the flow field and the temperature profile in the simulation domain. The airflow rate of a opening is calculated as the summation of flow per unit area as shown in equation.

\[
Q = \sum_{i=0}^{n} v_i \cdot A_i
\]

where:

- \( Q \) the volume flow rate of the opening, \( m^3/s \)
- \( v_i \) the velocity of mesh element \( i \) on opening surface, \( m/s \)
- \( A_i \) the area of mesh element \( i \) on opening surface, \( m^2 \)

Airflow Rates Comparison

The airflow rates of each of the openings computed from the airflow network model and the coupled CFD model are shown in Fig. 2, and air change rates for the 5 zones are shown in Fig. 4. The following observations were obtained:

- Airflow network model generally predicts smaller airflow rates for the openings.
- Airflow network model generates better airflow rates for openings on the south and east facade and internal openings, with difference ranging from −100% to −200% from the coupled model of about 2.5 m^3/s to 6 m^3/s.
- The difference between the airflow network model and the coupled CFD simulation for openings on the north facade and clear story openings are generally bigger (600% for a clear story opening from the coupled model of about 2 m^3/s, and −600% for an opening on north facade from the coupled model of about 1 m^3/s). The reason for the bigger difference on north facade is because that there is a very close adjacent building to the north of the building which makes the airflow pattern near the north facade more complicated, which is hard to capture with airflow network model. The percentage differences of airflow rate for openings on north facade range from 100% to 770% from the coupled model of about 1 m^3/s.
- Due to the differences in the predictions of airflow rates of all the openings, the resulting air change rates of all the zones are quite different. Especially for zone Z2, which has clear story openings, the airflow network generates relatively higher airflow rates compared to the coupled simulation, therefore, the air change rate prediction from airflow network model of zone Z2 is higher than that computed from the coupled model. Air change rates predicted by nodal model for other zones are generally lower than that predicted by the coupled model, as a result of the differences in airflow rate calculation of the openings.

CONCLUSION

A coupled simulation platform between nodal model and Finite Volume Method (FVM) model for advanced building thermal simulation was implemented and tested with a live retrofit building project at the Philadelphia Navy Yard.

From the study of airflow rates through openings under natural ventilation, it is found that Airflow network model in EnergyPlus generates same airflow rate for openings with same area, on the same facade of the same thermal zone. Airflow network model generally predict smaller airflow rates for the openings. Airflow network model generates better airflow rates for openings on the south and east facade and internal openings, with difference ranging from −100% to −200% from the coupled model of about 2.5 m^3/s to 6 m^3/s. Due to a very close adjacent building to the north of the building, the difference between airflow network model and coupled CFD simula-
Airflow Rate Comparison Between Coupled CFD Model and Nodal Airflow Network Model for Windows on South Facade

Airflow Rate Comparison Between Coupled CFD Model and Nodal Airflow Network Model for Windows on North Facade

Airflow Rate Comparison Between Coupled CFD Model and Nodal Airflow Network Model for Windows on East Facade

Airflow Rate Comparison Between Coupled CFD Model and Nodal Airflow Network Model for Clear Story Openings

Airflow Rate Comparison Between Coupled CFD Model and Nodal Airflow Network Model for Internal Doors

Figure 2: The flow rates comparison between nodal airflow network model and coupled CFD model for windows on the south (a), north (b) and east (c) facade. Airflow rates comparisons for the clear story (d) and internal doors (e).

Figure 3: The averaged percentage difference of airflow rate values during run period of June 1st to June 8th between values generated with nodal airflow network and coupled simulations.

Figure 4: The Air Change Rate (ACH) values for the 5 zones during run period of June 1st to June 8th between values generated with nodal airflow network and coupled simulations.

The Air Change Rates Comparison Between Airflow Network Model and Coupled CFD Model

The Percentage Difference of Flowrates Between Nodal Airflow Network Model and Coupled CFD Model

Airflow Network Coupled CFD Z1

REFERENCES


Chen, Q., X. Peng, and A.H.C. van Passen. 1995. “Pre-
diction of room thermal response by CFD technique with conjugate heat transfer and radiation models.” *ASHRAE Transactions* 101:50–60.


Wang, Liang, and Nyuk Hien Wong. 2009. “Coupled simulations for naturally ventilated rooms between building simulation (BS) and computational fluid dynamics (CFD) for better prediction of indoor thermal environment.” *Building and Environment* 44:95–112.


Zhai, Zhiqiang, Yang Gao, and Qingyan(Yan) Chen. 2004. “Pressure boundary conditions in multi-zone and CFD program coupling.” *1st IBPSA-USA Conference*. 836