2015

High Fidelity CFD Modeling of Natural Ventilation in a Solar House

Mirka Deza
Iowa State University, mdeza@iastate.edu

Baskar Ganapathysubramanian
Iowa State University, baskarg@iastate.edu

Shan He
Iowa State University

Ulrike Passe
Iowa State University, upasse@iastate.edu

Follow this and additional works at: http://lib.dr.iastate.edu/me_conf

Part of the Architectural Engineering Commons, Computational Engineering Commons, and the Fluid Dynamics Commons

Recommended Citation
http://lib.dr.iastate.edu/me_conf/175

This Conference Proceeding is brought to you for free and open access by the Mechanical Engineering at Iowa State University Digital Repository. It has been accepted for inclusion in Mechanical Engineering Conference Presentations, Papers, and Proceedings by an authorized administrator of Iowa State University Digital Repository. For more information, please contact digirep@iastate.edu.
ABSTRACT

Natural ventilation is an important factor in the design of sustainable buildings; it has the potential to improve air quality, while providing thermal comfort at reduced energy costs. Computational fluid dynamics (CFD) simulations provide comprehensive information on the internal flow pattern and can be used as a design tool. The present work offers insight of natural ventilation in a fully functional building, namely, the solar facility Interlock House in Iowa. Ventilation in the house is studied during summer months with some of its furniture included. The results show quantitative agreement between numerical simulations and experiments of vertical temperature profiles for each room. The temperature profile of the room with the inlet opening shows a more pronounced temperature variation. Flow patterns show higher velocities near the walls and marked flow circulation towards the opposite side of the building. The purpose of this work is to validate the numerical model that predicts airflow distribution for different configurations.

INTRODUCTION

Almost 40% of the energy usage in the US is utilized in buildings and about 48% of that usage corresponds to heating and cooling. Natural ventilation is an important factor in the design of sustainable buildings; it has the potential to improve air quality, while providing thermal comfort at reduced energy costs. Although the benefits of natural ventilation make it an attractive alternative, as wind and thermal energy are free resources, a good understanding of the physics of ventilation is necessary given the difficulty in being able to control such available resources [1]. In 1999, Linden [1] discussed the fluid mechanics of natural ventilation explaining the effects of wind- and buoyancy-driven flows due to temperature variations by exploring mixed and displacement ventilation. In the last decades many studies have been performed to better understand the flow behavior in natural ventilated buildings. Some of the methods used to study ventilation in buildings include analytical modeling, experimental models performed in small scale and full scale buildings, zone network models, and computational fluid dynamics (CFD). According to Chen [2], who summarized the methods utilized to predict ventilation performance, CFD was not only the most popular, but the most accurate and reliable modeling method. CFD is mainly to study natural ventilation, indoor air quality, and stratified ventilation, which is hard to predict with the other methods. Different CFD applications include commercial and residential buildings, health care, industrial, and agricultural facilities, as well as schools, and public transportation vehicles.

CFD simulations has extensively been used to study natural ventilation flows because it provides comprehensive information on the flow pattern inside buildings and parameters to calculate comfort and efficiency. CFD can also be used as a tool to explore different ventilation scenarios and determine optimal configurations, loads, and operation conditions. Bastide et al. [3] used a combination of zonal models and CFD to investigate thermal comfort in tropical climates and reduce air-conditioning usage. Many studies have focused on numerical models, especially to determine the appropriate turbulence model. Jiang and
Chen [4] compared full-scale experiments with CFD to study buoyancy-driven ventilation in a single-sided room; they determined that large eddy simulations (LES) models were more accurate than Reynolds-average Navier-Stokes equations (RANS). Further studies were performed by Allocca et al. [5] by increasing the number of heat sources in the room and stacking them as a three-story building. For this larger case, a RANS model, namely the re-normalization group (RNG) $k-\varepsilon$ was used; however, ventilation rates were underestimated. Evola and Popov [6] used the experimental results from the previous authors to test two turbulence model, $k-\varepsilon$ and RNG $k-\varepsilon$, for three different cases including one of cross-ventilation. They concluded that the RNG $k-\varepsilon$ model was in better agreement with the experiments. A realizable $k-\varepsilon$ turbulence model was also investigated and compared by Stavrakakis et al. [7] and no significant difference was reported. Hu et al. [8] investigated shear-stress transport (SST) $k-\omega$ turbulence model and found it predicted more accurate results than the previous models, not only around a building, but inside a cross-ventilated building. The authors later determined that for unsteady flows produced by fluctuations from the wind, LES models were more appropriate [9]. Ramponi and Blocken [10] also found SST $k-\omega$ turbulence model better than other RANS models, including standard $k-\omega$ and Reynolds stress model (RMS). Besides comparing turbulence models, these researchers also studied the impact of other computational parameters, such as, the size of the computational domain, the grid resolution, the inlet turbulent kinetic energy, the order of discretization scheme, and the level of iterative convergence in a cross-ventilated generic isolated building.

Most research performed in this area focuses on one room, the atrium or façade of a building, and most previous work validate their numerical model with a small-scale or a full-scale lab model of the building. There is some CFD modeling that has been used in full-scale buildings to study interior natural ventilation. Stokes et al. used CFD to predict natural ventilation flow in a historical building, the Viipuri Library, with the purpose of assisting in the renovation and restoration of the mentioned building [11]. The authors also modeled natural ventilation for different weather conditions in the Esherick House [12], a single-room house whose architectural design provides natural ventilation and lightning. Other CFD simulations have the purpose of studying natural ventilation in historical buildings to learn what techniques were used. Some of that work was performed by Balocco and Grazzini in the book deposit room of the old Marchese Building in Palermo and the left-wing summer apartments of the Pitti Palace in Florence, both in Italy [13].

The present work offers a unique insight of natural ventilation in a fully functional building with some of its furniture included. Not only accurate behavior of the airflow inside the building and its actual operation conditions are recorded experimentally, but whole building simulations are carried out to validate the model and analyze the airflow. Experimental data taken in the Interlock house during the summer were analyzed to determine a ten-minute period with uniform data and averaged to use as boundary conditions. For the numerical simulations, a buoyancy heat transfer steady-state model with RANS turbulence model, namely, SST $k-\omega$ was used to predict natural ventilation inside the house. The model was implemented in the open source program package Open Field Operation and Manipulation (OpenFOAM®). Simulations used High Performance computing (HPC) clusters with 128 and 256 processors for domain discretization ranging from 1.3 to 10 million cells. Vertical temperature profiles were used to establish an appropriate grid resolution. Resultant velocity and temperature distribution were analyzed and discussed.

**EXPERIMENTAL SETUP**

The Interlock house, is a solar facility built by Iowa State University faculty and students in 2009 for the U.S. Department of Energy Solar Decathlon. It is currently owned and operated by the Iowa Department of Natural Resources as a Conservation Activities Center at Honey Creek Resort State Park in Moravia, Iowa. The 74.3 square-meter house with a modular design is comprised of six components that interlock together. The house has four main rooms, kitchen, living room, bedroom, and bathroom as shown in Fig. 1. It also has a sunroom in the front with movable glass doors that make it possible to store thermal energy and also increase ventilation. There are a total of three windows located in the south and east walls and six clerestory windows located in the north walls.

The Interlock house is equipped with an extensive data acquisition and storage system, that includes 88 permanently installed sensors for temperature, humidity, internal air velocity and illumination and a full weather station collecting exterior so-
lar radiation, temperature, humidity, wind speed and direction. Data is collected every minute and stored in the data logger memory, which can be remotely accessed via Internet connection. In addition to the existing setup, a custom designed portable system for data collection, Mobile Data Acquisition System (MiDAS), has the ability to record dynamic airflow in the interior of the house. MiDAS measures temperature, multidirectional air velocity, and humidity and it can be adjusted for different scenario. MiDAS has been designed as a tool for accurately measuring stratification of temperature and airflow direction over a period of time, which provides a one-of-a-kind opportunity to precisely design an experiment in a full scale building using real-time data and nicely complements the existing data collection system of the house.

The current research studies natural ventilation around the house for the summer configuration with the sunroom glass doors open to the interior. The experiment was planned to validate a natural ventilation scenario with one main inlet at the south east window of the kitchen and six outlets at the clerestory windows of the north façade. The experiment was conducted during night time after air conditioning had been turned off for about 5 hours. There were 10 one-directional anemometers being used. Five of them were used at the inlet and the other five were distributed to five of the six outlets. Three of them were placed at the center of the inlet as one cluster measuring air velocity in three directions and two more were placed at 0.25 inches from the edges. At the outlets, anemometers were placed at the center of the window in the living room next to it. Fifty fast-response thermisters were used, two were placed at one inlet and one outlet and two more were placed at 0.25 inches from the edges. At the outlets, anemometers were placed at the center of the window in the living room next to it. Fifty fast-response thermisters were used, two were placed at one inlet and one outlet and 48 were mounted on the poles at the center of each room. Each pole has 12 thermisters distributed from 0 to 2.1 m height. Data sampling rate is 0.2 Hz and sensor accuracy is ± 0.1°C and ± 0.2% m/s for the thermisters and anemometers, respectively.

NUMERICAL MODEL

Equations for buoyancy driven flows are implemented in a C++ open source code Field Operation and Manipulation, OpenFOAM®. In OpenFOAM®, matter is represented as a continuum and uses a segregated iterative solution, that is, governing equations are in separate matrix equations and solved iteratively. The continuity equation for a compressible flow is:

\[
\frac{\partial}{\partial t} (\rho \mathbf{u}) + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = 0
\]  

Where \( \rho \) is the density and \( \mathbf{u} \) the velocity vector. For this work, simulations were initially run using steady-state equations and the solution was used as the initial condition for the transient simulation. For steady-state simulations, the partial derivatives with respect to time terms are set to zero. The momentum equation and energy equation have the form:

\[
\frac{\partial}{\partial t} (\rho \mathbf{u}) + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot (2\mu \mathbf{S} - \rho \tau) \tag{2}
\]

\[
\frac{\partial}{\partial t} (\rho h) + \nabla \cdot (\rho \mathbf{u} h) + \frac{\partial}{\partial t} (\rho k) + \nabla \cdot (\rho \mathbf{u} k) = \frac{\partial p}{\partial t} + \nabla \cdot (\alpha \nabla h) \tag{3}
\]

Where \( p \) is the pressure, \( S \) is the mean strain-rate stress tensor which corresponds to \( \frac{1}{2}(\nabla \mathbf{u} + \nabla \mathbf{u}^T) \), and \( \tau \) is the Reynolds stress tensor, which can be written as \( \mathbf{uu}' \). For the energy equation (Eqn. 3) \( h \) is the enthalpy, \( k \) is the kinetic energy, and \( \alpha \) is the thermal diffusivity. Similarly, for steady-state simulations, the first term on the left-hand side of the equations is set to zero.

The shear-stress transport (SST) model for the \( k - \omega \) turbulence model, known as \( k - \omega \) SST is based on the equations developed by Menter [14] and are as follows:

\[
\frac{\partial}{\partial t} (\rho k) + \nabla \cdot (\rho \mathbf{u} k) = \tau \nabla \cdot \mathbf{u} - \beta' \rho \omega k + \nabla^2 \left[ (\mu + \sigma_k \mu_t) k \right] \tag{4}
\]

\[
\frac{\partial}{\partial t} (\rho \omega) + \nabla \cdot (\rho \mathbf{u} \omega) = \alpha \rho S^2 - \beta \rho \omega^2 + \nabla^2 \left[ (\mu + \sigma_\omega \mu_t) \omega \right] + 2(1 - F_1) \rho \sigma_{\omega 2} k \nabla \cdot \nabla \omega \tag{5}
\]

Where \( \omega \) is the specific rate of dissipation of the turbulence kinetic energy, \( \mu \) is the dynamic viscosity, \( \mu_t \) is defined as \( \rho k/\omega \), \( F_1 \) is a blending function that is 1 near the wall and 0 away from the wall. The other terms, \( \beta' \), \( \beta \), \( \sigma_k \), \( \sigma_\omega \), and \( \sigma_{\omega 2} \) are constants.

Numerical Methodology

Equations are discretized using a finite volume method and cell-centered variables. SIMPLE and PIMPLE, a combination of SIMPLE and PISO, algorithms are used to couple pressure and velocity for steady-state and transient simulations. Specifics about the solvers, buoyantSimpleFoam and buoyantPimpleFoam can be found in the OpenFOAM® user guide [15]. Time derivatives use a first order Euler method while spatial derivatives are a blend of first- and second-order. Simulations were run using the CyEnce High Performance Computing cluster at Iowa State University. CyEnce is comprised of 240 servers each with 16 cores, 128 GB of memory and 40 Gbit interconnects. Other 24 nodes are similar, but contain 2 GPUs, and 24 more nodes contain 60 core accelerator cards.
### TABLE 1: NUMBER OF CELLS, PROCESSORS, CPU TIME, AND WALL TIME FOR THREE DIFFERENT MESH SIZES.

<table>
<thead>
<tr>
<th>Grid</th>
<th>Coarse</th>
<th>Medium</th>
<th>Fine</th>
</tr>
</thead>
<tbody>
<tr>
<td># cells (million)</td>
<td>1.3</td>
<td>5</td>
<td>10</td>
</tr>
<tr>
<td># unknowns (million)</td>
<td>9.1</td>
<td>35</td>
<td>70</td>
</tr>
<tr>
<td># processors</td>
<td>128</td>
<td>128</td>
<td>256</td>
</tr>
<tr>
<td>CPU time (h)</td>
<td>47.8</td>
<td>93.2</td>
<td>96</td>
</tr>
<tr>
<td>wall time (h)</td>
<td>48.5</td>
<td>98.8</td>
<td>132.4</td>
</tr>
</tbody>
</table>

### Domain and Boundary Conditions

The present research was conducted to validate a buoyancy heat transfer numerical model with k-ω-SST turbulence model that will predict the natural ventilation flow around a four-room house and compare the results with the real-scale facility. The simulation uses a 3D model of the house interior, that is, only the air is being modeled and wall temperatures are taken as boundary conditions. Sizes of windows have been reduced using the approximated projected horizontal area. All the windows in the house are fully open by tilting a windowpane 45°. The house dimensions are approximately 11.8 m × 5.3 m with inclined ceiling that goes from 2.5 m to 4.7 m. Figure 1 shows the geometry of the interior of the Interlock house. To determine an appropriate number of cells three different cases were tested with the experiment with 1.3, 5, and 10 million cells. Table 1 shows the number of processor, number of unknowns, CPU time and wall time for each case. These times include steady-state and transient simulations. Coarse grid simulation ran for 50,000 steps, and medium and fine, for 22,500 and 15,900 steps, respectively before running as transient. All the transient simulations run for at least 70 seconds. Contrary to the steady-state simulation, transient simulations yield very low residuals achieving convergence.

Inlet velocity, wind direction, and temperature in the room were used to determine a 10-minute period with the smallest variation by calculating the standard deviation of these parameters. All measurements were averaged over that period and used in the simulations as boundary conditions in the walls and temperatures in the poles as data for validation. The incoming air for the first case is 1.2 m/s normal to the inlet at a temperature of 294 K (21°C).

### CASES AND RESULTS

A grid resolution study is performed to determine the appropriate number of cells for the Interlock house numerical simulations. The vertical temperature profile at each room for the three mesh sizes is compared with experimental data to establish a sufficient grid size for the CFD model. The selected grid size will allow us to study the flow and make further predictions. Figure 2 shows temperature versus height for (a) bathroom, (b) bedroom, (c) living room, and (d) kitchen. Bathroom and bedroom temperatures profiles for the fine grid perfectly match the corresponding temperature profiles obtained from the experiments. Temperature profile for the fine grid in the living room is very close and in good agreement with experiments. In the kitchen, all simulations predict lower than measured temperatures below 1 m height. This might be due to limitations in the boundary conditions that results in a slightly different temperature distribution; however temperatures over 1 m height using the fine grid are in good agreement with the experimental data.

Temperature and velocity magnitude contour plots of selected planes across the rooms are shown from Figs. 3–5, all of them using the same scale. The planes, as indicated in Fig. 1 with red lines, go from west to east. The first plane on the west side crosses both the bathroom and the bedroom of the Interlock house. It is observed that the temperature in the bathroom is lower than the one in the bedroom, especially toward the wall.
that separates both rooms. The bedroom on the other hand shows lower temperatures toward the front of the house. In both rooms, the temperature is around 294 K (Fig. 3 (a)). The velocity distribution (Fig. 3 (b)) shows higher velocities toward the walls and one zone of flow circulation towards the center of the bedroom, being the highest around 0.25 m/s. The bathroom initial temperature was low and airflow cause the temperature to rise as can be observed in the comparison of the temperature and velocity distribution. The bedroom, on the other hand, started with a higher temperature and airflow has caused the temperature to decrease.

At the center of the house, is the living room. Contour plots (Fig. 4) show higher airflow near the walls, especially in a region in the ceiling that separates the sunroom and living room. There is another region of noticeable airflow in the center of the living room. No much variation is observed here as the flow mostly moves from east to west, perpendicular to the plane.

The kitchen (Fig. 5), the room where the main inlet is located, clearly shows the region of the inlet with a plume-like shape of lower temperature and high velocity. The air entering the room in combination with air flowing down from the top of the room forces the airflow back and around the room close to the walls. Due to this circulation pattern, the small space over the kitchen countertop is kept at its original higher temperature and almost no airflow is observed here.

Contour plots of these planes containing vector fields and streamlines of incoming air are presented in Figs. 6 and 7, respectively. The vector field confirms the air flowing at higher velocity near the walls. In the living room airflow can be seen going from east to west in the from wall and mostly top to bottom in the back wall. Air goes all the way to the bedroom and comes back toward the living room, as is observed in the vector field in the bedroom. Air does not leave through each of the clerestory windows; a small flow enters the house into the living room and kitchen, only leaving the house in the bathroom at the opposite side from the inlet. A more clear depiction of the circulation pattern can be observed in the streamlines. The incoming air tends to circulate toward the front of the house. The circulation pattern towards the back walls is mostly due to the incoming air from the clerestory windows.

Preliminary results of ventilation in the fall season using a coarse grid are presented as streamlines in Fig. 8 and further study needs to be performed. In this additional case, the inlet is in the bedroom and one outlet is in the kitchen. A total of sixty thermistors measure temperatures at each wall, floor, and ceiling. These preliminary results show more circulation around each room before going to the next, especially in the bedroom and living room. Having more thermistors is providing more accurate boundary conditions and therefore a more realistic pattern of the airflow as buoyancy effects are better modeled.
The research presented in this paper shows natural ventilation flows in a full scale building with some furniture in a real environment. The advantage of having accurate real time information is that it can be used as a design tool for architects to determine the optimal configuration of a current operational building for specific sets of external and internal conditions. CFD simulations could also have an impact in the design of new facilities by allowing the selection of optimal locations or orientations of a building and providing guidelines to determine the size and location of the openings to maximize the use of natural ventilation.

**CONCLUSIONS**

A solver for buoyant, compressible, turbulent flow for heat-transfer and ventilation implemented in OpenFOAM® with $k-\omega$ SST turbulence model was used to model natural ventilation in the Interlock house during summer with one main inlet and all six clerestory windows opened. Experiments were conducted to gather information for boundary conditions and validation. Temperatures at 11 different points were taken at the walls, velocity vectors and temperature were measured at two openings and used as boundary conditions, while temperature distributed vertically in poles located in the center of the rooms were used for model validation.

A grid resolution study shows that a fine grid of about 10 million cells is in better agreement with experimental data. Initial results are obtained using a steady-state model, but convergence is not fully achieved, as residuals are not as low as expected. Those results are used as initial conditions for transient model, where residuals are very low, showing convergence. Temperature distribution shows masses of air circulating at different temperature than air temperature where circulation is poor. Velocity distribution, as observed in the contour plots indicates that air moves faster near the walls, which is clearly displayed in the vector field and streamlines. Streamlines also illustrate how the incoming air moves around the front of the house, while the flow toward the back is most probably generated by incoming air from four of the clerestory windows. The incoming air from the clerestory windows in the kitchen is responsible for the flow circulation observed over the inlet plume, where incoming air is pushed back and around the front wall before going around the ceiling and to the other rooms.

The results of this research show quantitative agreement between real scale experiments and numerical simulations. The current model will be used to simulate other inlet/outlet configurations. It is the objective of the authors to use the numerical model to study the optimal configuration for natural ventilation under different weather conditions in existing facilities and to use it as a design tool to plan for future buildings.

**ACKNOWLEDGMENT**

This material is based upon work supported by the National Science Foundation under Grant Number EPS-1101284. Any opinions, findings, and conclusions or recommendations ex-
pressed in this material are those of the author(s) and do not necessarily reflect the views of the National Science Foundation. The authors would like to thank the financial support of the Iowa Energy Center for partially funding the present work through grant No. 14-011-OG. The authors would also like to thank students Songzhe Xu, Esdras Murillo, Ryan Everly, Kelsey Fleenor, Shuaibu Kenchi, and Evan Jeanblanc for preparing and installing the anemometers and thermistors, and for programming the data loggers. Thanks to Dr. Alberto Passalacqua and Anthony Fontanini for their insight with the OpenFOAM® numerical model, and Erika Deza for her assistance in the preparation of the internal domain.

REFERENCES