Applied & Computational Mathematics

Sapkota et al., J Appl Computat Math 2015, 4:6
DOI: 10.4172/2168-9679.1000276

Research Article Open Access

Modeling of Indoor Air Flow Distribution in a Natural Cross-ventilated Kitchen

Sapkota BP[1], Uprety KN[1] and Gokhale S[2]

[1]Institute of Science and Technology, Tribhuvan University, Kathmandu, Nepal

[2]Centre for the Environment, Department of Civil Engineering, Indian Institute of Technology, Guwahati, India

Abstract

Most of rural population in developing countries depends on the biomass energy for cooking and heating purposes. Women in this sector spend large part of their daily time in the kitchen cooking the food and suffering from the contaminants emitted from the biomass cook stoves. This study focuses on the numerical modeling of indoor air pollutant in a single room Nepali kitchen by solving the Navier-Stokes equations to analyze velocity profile and temperature distribution throughout different sections of the kitchen to depicted the pollutant paths. The study explored the fluid flow profile, temperature profile based on the changing parameters such as inlet velocity, number and position of ventilation proper positioning of the ventilation to minimize the effect of pollutant to the person working in the kitchen.

Keywords: Indoor air pollution; Navier-stokes equation; Numerical modeling; Natural ventilation

Introduction

Air pollution is the presence of one or more contaminants in outdoor atmosphere in a sufficient quantity and duration to cause them to be injurious to human health and welfare and animal and plant life and to interfere with the enjoyment of life and property. Particulate matter (PM) is one of the most important pollutant of the air which is a complex mixture of extremely small solids and liquid droplets. It is made up of a number of components, including acids, organic chemicals, metals, and soil or dust particles. Their properties are often categorized according to aerodynamic particle diameter in micrometers. Particles with aerodynamic diameter between 20-50 micrometers (μm) are the total suspended particles (TSP) that can be filtered by nose and mouth so they have less health effects. Particles with aerodynamic diameters less than 10 micrometers are called PM_{10} and may reach the upper part of the airways and lung. Particles with aerodynamic diameters 2.5 micrometers and smaller are regarded as more dangerous because they penetrate more deeply into the lungs and may reach the alveolar region. Taking width of human hair as 100 μm, the width of bacteria, road dust, coal dust, viruses and vehicle emissions are 10, 5, 2, 0.4 and 0.2 μm respectively. Similarly ultra-fine particles are the particles with aerodynamic diameters 0.1 micrometers and smaller [1].

Indoor air pollution(IAP) is a global burden of the people in the world. The people of the developing countries are especially victimized due to the different diseases related to IAP. About 90 percent of the time of the people spent in indoor environment either inside the office, or in the vehicles or inside the home. The accumulation of contaminants that comes primarily from inside the building can constitute a potential health hazard in microenvironments where people spend most of their time indoors. This research investigates the distribution pattern of the contaminants inside a kitchen and the analysis of the flow driven by the buoyancy in a naturally ventilated room in cross ventilated windows in the kitchen.

Simulation has been made to compare the results in different parameters of indoor air pollution and contaminant distribution. This numerical computational model of the fluid flow guides the concentration behavior in enclosures which will be helpful tool to predict the velocity fields, distribution of the pollutant level generated inside the kitchen and the thermal comfort to the people living inside the kitchen.

Literature Review

In 1960 Brief proposed single graphs to determine transient concentration for pollutants in indoor setting that is based on exponential decay as a function of time. Turk purposed a general equation for calculating concentration in a chamber that included both exterior and interior source and the removal effect of pollutants by air treatment system in 1963. Jones and Fagon used Turkes Equation to calculate carbon mono-oxide (CO) concentration from cigarette smoke in 1974. In 1974 Nielson used numerical predictions of indoor airflow and worked mainly on two-dimensional, steady and isothermal flows. Even though his two-dimensional results are not very useful for engineering applications, the methods he used showed a very strong potential for solving practical airflow problems in a room. Since the end of the 1970's, three-dimensional computations of turbulent airflow in a room became more and more popular because some powerful computers became available. Many researchers validated their computational results with experiments.

Lu et al. used computational fluid dynamics(CFD) to simulate airflow/temperature in a room to track pollutant dynamics and found that the CFD results correlated reasonably well with measured experiments. Sinha et al. also used finite volume CFD to compare discrete vent configuration cases, but also ran simulations for different fixed Reynolds number Re and Grashof number Freire et al. studied the problem of optimizing thermal comfort and energy savings

***Corresponding author:** Sapkota BP, Institute of Science and Technology, Tribhuvan University, Kathmandu, Nepal, Tel: 97714331969; E-mail: buddhisapkota@gmail.com

Received November 07, 2015; **Accepted** December 21, 2015; **Published** December 24, 2015

Citation: Sapkota BP, Uprety KN, Gokhale S (2015) Modeling of Indoor Air Flow Distribution in a Natural Cross-ventilated Kitchen. J Appl Computat Math 4: 276. doi:10.4172/2168-9679.1000276

using model-based predictive control where their controlled input was to apply power to the HVAC device [2]. Considerable progress has been made recently in developing mathematical models for predicting pollutant concentration in air [2]. In 1980 Ishizu examined experimentally the inclusion of mixing factor into these models and Repace and Lawrey also developed a modification of the Turk equation incorporating a mixing factor. Trynor [3] studied the effects of ventilation on residential air pollution, due to emission from a gas-fired range in 1982 and showed that the range hood is the effective means of the removing pollutant from gas fired range; removal rates varied from 60 to 87 [3]. Results of a pilot study in four Indian villages of personal exposure to TSP and particulate benzo(a)pyrene (BaP) of women cooking on simple stoves using traditional biomass fuels are presented together with socioeconomic and fuel-use determinations in 1983 by Kirk [4]. In 1996 Ott et al. showed theoretically that a mathematical trend correlation term should be incorporated into the time averaged version of the model to make it exact. Neil E Klepeis [5] studied on validity of the 'uniform mixing' assumption: determining human exposure to environmental tobacco smoke, environmental health perspectives.

Hensen [6] demonstrated as a merit and drawbacks of various computer modeling approaches for HVAC design and performance prediction. He described the three level of room air modeling: fully mixed zones approach, intermediate approach and flow field approach. He concluded that whole body thermal comfort, local discomfort, ventilation efficiency, contaminant distribution are the potential modeling approach on CFD. He pointed out some of the future works including integration of CFD in general building energy simulation for third approach, prediction of thermal comfort as affected by the flow and temperature field within a room. Zhang [7] estimated CO concentrations and exposures in a well-mixed, hypothetical village kitchen using measured CO emission factors and published typical values for house, fuel and activity parameters. The estimate was made to compare CO exposure potentials for a range of no-flue fuel/stove combinations. Based upon this quantitative comparison, recommendations were made on CO exposure reduction by fuel/stove switching. If their house conditions and activity patterns are comparable to those typical values assumed in our estimation, people using biomass and coal cook stoves could have daily CO exposures greater than the exposure equivalents of health-based national standards and WHO guidelines. Wayne [8] developed a mathematical model for predicting indoor air quality from smoking activity. In 2001 ICMR bulletin [9] of India had published a research on the use of biomass fuel responsible with acute respiratory tract infections in children, chronic obstructive lung diseases, and pneumoconiosis in the residents of Ladakh villages. Chen [10] and Srebric [11] has developed a new model to assess building shape design, to evaluate effectiveness of natural ventilation in buildings to model volatile organic compound (VOC) emission from building materials and calculated indoor air environment parameter.

Methodology

Mathematical model

The study of three-dimensional incompressible flow of air as a multi-component fluid includes dry air and contaminant, the fluid properties vary according to ideal gas model and therefore, it accounts for the buoyancy associated to the natural convection of the heated fluids. The physical model is built by using design modeler. Then, the Reynolds-Averaged Navier-Stokes equations are prescribed along with mass, energy, ideal gas and conservation of species equations, with the added closure 2-equation enhanced k - ε turbulence model. The latter is the combination of the classical k - ε model and Launder-Spalding wall function approach. Finite volume method is used to discretize the governing equations, continuity, momentum, transport and energy equations [12,13].

The fluid finite element based finite volume model depends on flow velocity as the variable. In turbulent flow, the relative motion of the fluid in the boundary layer generates flow disturbances in the form of vortices or eddies. As the flow rate increases so does the amount of turbulence. There is a continuous transfer of energy from the main flow into large eddies, and from the large eddies into smaller eddies, which dissipate most of the energy. This process occurs in a narrow strip inside the boundary layer, in the neighborhood of the wall. This energy dissipation produces large kinetic energy losses in the fluid. As the fluid molecules in the vortices go from locations of higher kinetic energy to regions of lower kinetic energy, i.e., from near the edge of the boundary layer to near the wall, the kinetic energy of the fluid is converted into heat and potential energy in the form of pressure.

Modeling approach

The proposed model room of dimensions 3.2 m × 3.5 m × 2 m has been taken as a typical model kitchen. For room airflow simulation, the present investigation used the CFD technique to solve a set of partial differential equations for the conservation of mass, momentum, energy, and species concentrations. These equations govern flow, heat and mass transport in a room. Since airflow in the room is turbulent, the standard k - ε method is used. The governing equations can be closed with appropriate thermo-fluid boundary conditions at all the boundaries such as air inlets, outlets and wall surfaces. The values of velocity, temperature, kinetic energy, the dissipation rate of kinetic energy, and species concentration were set at the boundaries. To solve the unsteady case ANSYS was used in the present investigation with the corresponding boundary conditions. The program discretized the indoor space into non-uniform computational cells, and the discrete equations were solved with the SIMPLE algorithm.

Governing equations

The airflow, convective heat transfer, and species dispersion in the indoor environment are controlled by the governing equations of mass, momentum and energy (Navier-Stokes Equation), which can be expressed in a common form as equation.

$$\frac{\partial(\rho\phi)}{\partial x} + \frac{\partial(\rho u_j \phi)}{\partial x_j} = \frac{\partial}{\partial x_j}(\Gamma_\phi \frac{\partial \phi}{\partial x_j}) + S_\phi \tag{1}$$

The variable φ represents any of the predicted quantities such as air velocity, temperature, or species concentration at any point in the three-dimensional space. The equation refers to the change in time of a variable at a location is equal to the amount of the variable flux (i.e., momentum, mass, thermal energy).

The mathematical model is based on the set of governing equations and respective boundary conditions. The main premises are: steady state, Newtonian fluid, incompressible and turbulent flow. Therefore, the following favr-averaged governing equations and boundary conditions were prescribed: mass conservation; Navier-Stokes equations; energy conservation; constitutive relations between p, T and H; species conservation; and k - ε turbulence model with wall functions. These equations are shown as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0 \tag{2}$$

$$\frac{\partial(\rho u_i)}{\partial t}+\frac{\partial(\rho u_i u_j)}{\partial x_j}=-\frac{\partial p}{\partial x}+\frac{\partial}{\partial x_j}[\mu(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i})]+\rho g_i \quad (3)$$

$$\frac{\partial(\rho H)}{\partial t}+\frac{\partial(\rho u_i H)}{\partial x_i}=\frac{\partial}{\partial x_i}[\frac{k}{c_p}\frac{\partial H}{\partial x_i}]+S_H \quad (4)$$

where, u_i is the velocity component (u, v, w), p is the pressure, H the enthalpy and S_H a source term. The diffusion term is indicated by the kinematic viscosity μ, the thermal conductivity k and the specific heat c_p. The time is indicated with t, x_i is the coordinate axis (x, y, z), ρ is the density and g_i is the gravitational acceleration.

Boundary conditions

At the inlet a inlet velocity is specified. At walls, wall functions are used to calculate the wall shear stress. The CO concentration, released from kitchen is appropriately modeled as mass source. The air, which is entered in the kitchen is considered clean. Both velocity and temperature boundary conditions are used. We assume that air flow velocity on the boundary can be characterized by mixed Dirichlet (essential) and Neumann-type (natural) conditions. The piecewise-connected portion of the boundary with essential conditions consists of walls and velocity controlled inlets, disjoint from which is the piecewise-connected portion of the boundary with natural conditions. The boundary is connected and is the union of these two sets. This specification of velocity is reasonable for the one-room problem where the velocity boundary of a room is mostly surrounded by walls. As a fluid particle grows in proximity to a rigid wall, the greater will be the influence of shear forces from the wall so that in the limit, the velocity will theoretically be zero. Everywhere in the solution region ρ, u and T are given at time t=0. No slip boundary conditions $u=u_{wall}$ is applied on solid walls with $T=T_{wall}$ (fixed temperature) is used in the wall with the temperature flux at the solid wall and is taken as zero.

Discretization

In the descretization process the PDE system is transformed into a set of algebraic equations where we perform some important tasks like mesh generation (decomposition into cells/elements) which may be structured or unstructured, triangular or quadrilateral; CAD tools and grid generators (Delaunay, advancing front); mesh size, adaptive refinement in interesting flow regions. The task of space discretization is the approximation of spatial derivatives which can be done in different ways such as finite differences, finite volumes, finite elements methods; taking into consideration of high- vs. low-order approximations (Figure 1). Time discretization is a process of approximation of temporal derivatives which can be done in different ways such as explicit vs. implicit schemes, stability constraints; local time-stepping, adaptive time step control (Figure 2).

The set of governing equations are discretized and solved using the finite volume method(FVM) on a spatially rectangular computational mesh refined locally at the solid/fluid interfaces and in specified fluid regions where high gradients are expected. The FVM grants a conservative discretization of the governing equations, with spatial derivatives (fluxes) approximated with second-order upwind on the SIMPLE algorithm. Resulting linearized algebraic system of equations is solved by iteration.

Results

The simulation results show that the proper design of an efficient ventilation system provides better quality of air to the occupants of the buildings which in turn leads for the quality and healthy life to the people exposed to the pollutants.

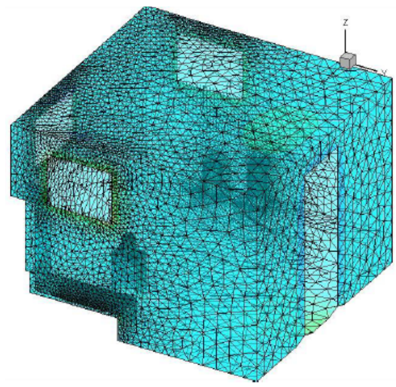

Figure 1: Mesh of the kitchen.

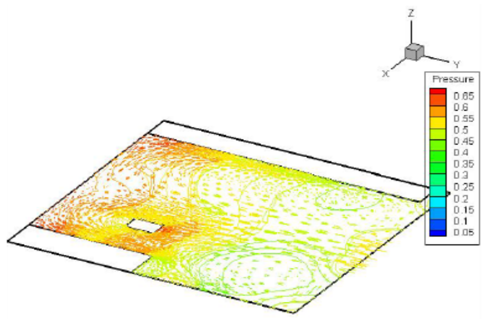

Figure 2: Pressure at the level of improved cookstove for two cross windows given in a z-plane.

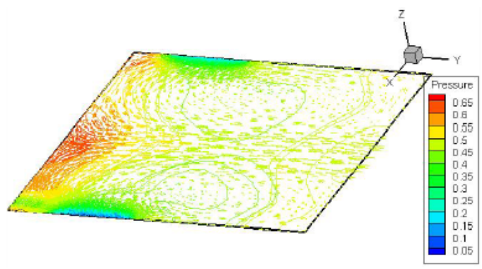

Figure 3: Pressure at the buttom level of window represented by z-plane.

Our study explored the distribution of indoor air inside a kitchen. Figure 2 shows the pressure distribution at the level of improved cook stove for the case of two cross windows given in a z-plane. We can observe the high pressure on man and the area opposite to the door and in between the cross ventilated windows. Figure 3 show the contours of pressure that indicates the distribution of pressure at the bottom level of window represented by z-plane. Here we can see the less pressure in the window level compared to the other region. The pressure is high on the opposite wall to that opposite of the door. Figure 4 represents the pressure at middle level of window represented by z-plane. This also indicates the ambient pressure in the window and more in between to windows. Figure 5 represents the pressure at level of head of person represented by z-plane and on the water harvesting pots. In Figure 6 we can see the path followed by the particles starting from the door. We can see the vortex formation in both sides of the path where the smoke

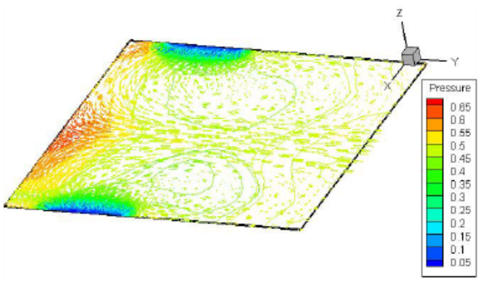

Figure 4: Pressure at middle level of window represented by z-plane.

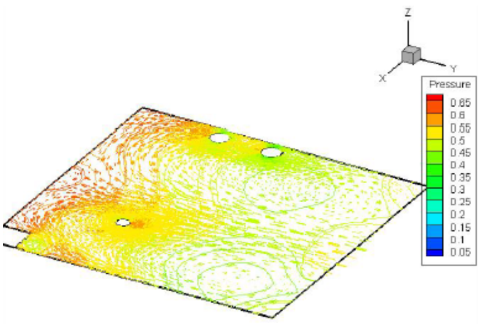

Figure 5: Pressure at level of head of person represented by z-plane.

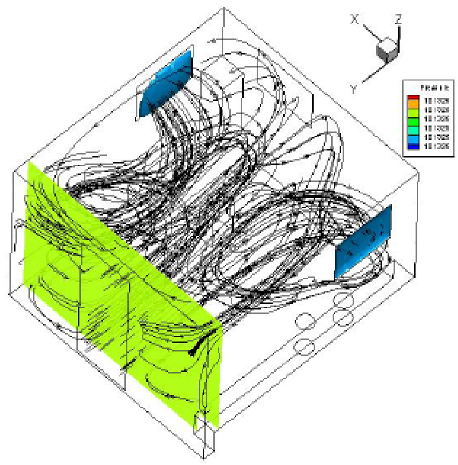

Figure 6: Path followed by particles in the kitchen.

is trapped. So the proper position of the person sitting in the kitchen could be adjusted in such a way that the she/he should avoid the place of vortex formation.

To validate the model, the measurement of the CO level along with the humidity and temperature is taken in a kitchen near by Indian Institute of Technology, Guwahati (IITG). The data collected are in good agreement with the simulation done with the ANSYS Fluent. A portable CO monitor was used to measure CO concentration in a typical kitchen. An amperometric monitor of CO sensor of model HD37AB17D with two-electrode sensor was used that detects the electrons given up by the CO molecules when they are oxidized to CO_2. The monitor has capacity to record data at given interval time stating from 3 seconds. Inside the kitchen measurements were performed from before starting the kitchen to after finishing it. Measurements of the CO after cooking till it decreases to zero are also taken. Two monitoring sites were selected to provide examples of typical workplace areas. Measurements were performed at a height of 1.5 m from the floor. From the continuous readings, instantaneous readings were stored for every 3 second by the data log system.

Conclusion

The research investigates the dispersion of indoor pollutants in the case of two open windows giving cross ventilation. We have validated the model by measuring CO levels at different kitchen. The onsite measured data has a good agreement with the numerical results. The pressure distribution at the level of improved cook stove for the case of two cross windows shows the high pressure on the area opposite to the door and in between the cross ventilated windows. We can see the vortex formation in both sides of the path where the smoke is trapped. So the proper position of the person sitting in the kitchen could be adjusted in such a way that the she/he should avoid the place of vortex.

References

1. Clean Air Initiatives (CAI) for Asian Cities-fact Sheet No. 2 pp: 1.
2. William NW (1989) Mathematical modeling and control of pollutant dynamics in indoor air. California Institute of Technology, Pasadinia California pp: 16.
3. Trynor GW, Apte MG, Dillworth JF, Holowwell CD, Sterling AM (1982) Effects of ventilation on residential air pollution, due to emission from a gas-fired range. Environmental International 8: 447-452.
4. Smith KR, Aggrawala L, Dave RM (1983) Air Pollution and Rural Biomass Fuels in Developing Countries: A Pilot Village Study in India and Implications for Research and Policy, Atmospheric Environment. Pergamum Press Ltd. 17: 2343-2362.
5. Klepeis NE (1999) Validity of the uniform mixing assumption: Determining human exposure to environmental tobacco smoke. Environmental Health Perspectives 107: 357-363.
6. Hensen MJL, Hamelinck MJH, Loomans MGLC (1996) Modeling approaches for displacement ventilation in offices. University of Tokyo, Yohohama pp: 1-8.
7. Zhang Smith KR, Uma R, Ma Y, Ma Kishore VVN, Lata K, et al. (1999) Carbon monoxide from cook stoves in developing countries: Exposure potentials. Global change Science, Chemosphere pp: 367-375.
8. Wayne RO (1999) Mathematical models for predicting indoor air quality from smoking activity. Environmental Health Perspectives pp: 375.
9. ICMR (2001) Indoor air pollution in India - A major environmental and public heath concern, Bulletin, based on numerical simulation of room airflow. University of Tokyo pp: 22-38
10. Chen Q (1988) Indoor air flow, air quality and energy consumption of buildings. Technische Universiteit Delf pp: 19-135.
11. Chen Q, Srebric J (2000) Application of CFD tools for indoor and outdoor environment design. International Journal on Architectural Science 1: 14-29.
12. Raju BSN (2005) Fundamentals of air pollution. Oxford and IBH Publishing Company pp: 1-2.
13. Sparks LE (2004) Indoor quality modeling, indoor air quality hand book. The McGraw-Hill Companies, USA pp: 58.