# REAL TIME AIRFLOW SIMULATION IN BUILDINGS

Wangda Zuo, and Qingyan (Yan) Chen†

*School of Mechanical Engineering, Purdue University, West Lafayette, USA*

## ABSTRACT

Real time flow simulation is crucial in emergency management in buildings, such as fire or accidental release of chemical/biological agents. Proper measures can be taken to minimize casualties with correct and timely prediction of the spread of the fire or contaminants. Although the traditional CFD simulation in buildings is accurate, it is too time consuming. Multizone flow modeling is fast, but its accuracy is poor. Therefore, it is very necessary to develop a new method that is faster than the traditional CFD, but more accurate than the multizone modeling.

Recently, the modified semi-Lagrangian method based on Navier-Stokes equation has been used for flow simulation. This method is unconditionally stable and can use a larger time step than traditional CFD. The method has been successfully used in computer game industry and in computer graphic science. However, the results are only virtually real and are not rigorously validated. This investigation used the method to systematically study three basic flows in buildings and compared the numerical results with the corresponding experimental data or direct numerical simulation data from the literature. The results conclude that it is possible to conduct flow simulations faster than real time by using the method, although some discrepancies exist between the numerical results and the data.

## KEYWORDS

CFD, Real Time, Semi-Lagrangian method, Fast Fluid Dynamics

## INTRODUCTION

Fire or accidental release of chemical/biological agents in buildings happens occasionally. In such emergent situations, quick prediction of the smoke or contaminant transport is crucial for proposing measures to minimize casualties. The prediction should be not only accurate and informative, but also faster than the real time.

Unfortunately, current modeling technologies cannot meet such requirements. Either their computing speed is too slow or their accuracy is too poor. For example, Computational Fluid Dynamics (CFD) by large eddy simulation (LES) of airflow and contaminant transport in a building demands an impractically large computer capacity (tens of Gb memory) and long computing time (weeks). Although CFD simulations using unsteady Reynolds averaged Navier-Stokes equations (URANS) are much faster than the LES, it still takes a desktop several hours to a few days to compute the airflow and contaminant transport in the building. On the other hand, by assuming the flow in a room is uniform, multizone flow network models need little computing time (a few seconds) (Wang 2007). However, the homogenous assumption of airflow in each room does not provide informative results for emergency management. Therefore, it is necessary to develop a method that is faster than the CFD, but more accurate and informative than the multizone modeling.

Weather forecast requires quick and accurate calculation of air motion and temperature of the atmosphere. By treating the linear terms responsible for gravitational oscillations in an implicit manner, Robert et al. (1972) proposed a semi-Lagrangian scheme. This scheme can increase the time step size

† Corresponding Author: *Tel: + 1 765 496 7562, Fax: + 1 765 496 7534*
*E-mail address: yanchen@purdue.edu*

by about six times at little additional cost and without degrading the accuracy of the solution. Applying the semi-Lagrangian approach, Staniforth and Cote (1991) calculated flow for weather forecast and Stam (1999) and Harris (2003) simulated fluid motion in computer games and achieved plausible results on real time. To distinguish the differences from traditional CFD, the method using semi-Lagrangian approach is named as “Fast Fluid Dynamics” or FFD.

We have attempted to use the FFD predicting indoor airflows (Zuo and Chen 2007). By comparing the computed results with corresponding data on indoor airflow from the literature, our results show that that FFD could predict such flows with reasonable accuracy and the simulations were faster than real time. However, our early work was for isothermal flow with uniform grids so that very fine grids were used for some cases. This investigation extended the simulations to non-isothermal airflows for indoor environment and developed our code further with non-uniform grid meshes. The results are reported in this paper.

## SCHEME OF FAST FLUID DYNAMICS

Before reporting the results, this section presents the basic equations and numerical techniques used by FFD. The FFD solves Navier-Stokes equations for incompressible fluid:

$$\frac{\partial U_i}{\partial x_i} = 0 , \tag{1}$$

$$\frac{\partial}{\partial t} U_i = -U_j \frac{\partial}{\partial x_j} U_i + \upsilon \frac{\partial^2}{\partial x_j^2} U_i + \frac{\partial}{\partial x_i} P + f_i , \tag{2}$$

where $U_i$ and $U_j$ are fluid velocity components in $x_i$ and $x_j$ directions, respectively; $\upsilon$ is kinematic molecular viscosity; $P$ is pressure; and $f_i$ is forces, such as buoyancy force. Applying the Euler approach to the scalar variables (such as contaminant concentration and air temperature), the state equation of the contaminant concentration or air temperature is:

$$\frac{\partial}{\partial t} C = -U_j \frac{\partial}{\partial x_j} C + k \frac{\partial^2}{\partial x_j^2} C + S , \tag{3}$$

where $C$ is contaminant concentration or air temperature; $k$ diffusivity; and $S$ source. In each time step, the FFD solves the Navier-Stokes equations (1) and (2) in four stages:

$$U_i^{(0)} \underset{\text{add force}}{\rightarrow} U_i^{(1)} \underset{\text{diffuse}}{\rightarrow} U_i^{(2)} \underset{\text{advect}}{\rightarrow} U_i^{(3)} \underset{\text{project}}{\rightarrow} U_i^{(4)} . \tag{4}$$

At the first stage, the FFD simply adds the force term in equation (2) as:

$$U_i^{(1)} = U_i^{(0)} + \Delta t \, f_i , \tag{5}$$

where $\Delta t$ is the time step. The second stage is to solve the diffusion term in equation (2) through a first order implicit scheme:

$$\frac{U_i^{(2)} - U_i^{(1)}}{\Delta t} = \upsilon \frac{\partial^2 U_i^{(2)}}{\partial x_j^2} . \tag{6}$$

By applying the implicit scheme, the simulation is always stable even when the Courant number is much large than 1. The third stage is to solve the advection term in equation (2):

$$\frac{\partial U_i^{(3)}}{\partial t} = -U_j^{(2)} \frac{\partial U_i^{(3)}}{\partial x_j} , \tag{7}$$

with a semi-Lagrangian approach (Courant et al. 1952):

$$U_i^{(3)}(x_j) = U_i^{(2)}\left(x_j - \Delta t\, U_j^{(2)}\right), \tag{8}$$

where $U_i^{(3)}(x_j)$ is $U_i^{(3)}$ at location $x_j$ = ($x_1$, $x_2$, $x_3$). However, the $U_i^{(3)}$ does not satisfy the continuity equation (1). Hence, the last stage is to correct $U_i^{(3)}$ by a pressure-correction projection scheme (Chorin 1967). The projection operation ensures the conservation of mass and it solves a Poisson equation for pressure:

$$\frac{\partial^2 P}{\partial {x_i}^2} = \frac{\partial U_i^{(3)}}{\partial x_i}\,. \tag{9}$$

The velocities are then corrected by

$$U_i^{(4)} = U_i^{(3)} - \partial P / \partial x_i\ , \tag{10}$$

where $U_i^{(4)}$ is the velocity satisfying the continuity equation (1). A similar approach can be applied for scalar variable state equation (3) for comtanimant concentration or air temerpature except the projection stage.

## RESULT ANALYSES

This investigation studied three typical indoor airflows: (1) a fully developed flow in a plane channel; (2) a natural convection flow in a tall cavity; and (3) a forced convection flow in a ventilated room. The flows represent the most basic elements of flows found in buildings.

### Fully Developed Flow in a Plane Channel

Flow through a corridor in a building is similar to that in a plane channel. Therefore, this study selected a fully developed flow in a plane channel as a test case for the FFD.

Based on wall shear velocity, $U_\tau$, and the channel half-width, $H$, the flow Reynolds number studied is $Re_\tau$ = 180. Kim et al. (1987) did direct numerical simulation (DNS) for this flow and their data were used as reference.

The FFD simulation was carried out with 64 × 32 non-uniform grids. Figure 1 compares the normalized mean streamwise velocities obtained by the FFD with the DNS data. The FFD can capture the main shape of the velocity profile, although it under-predicts the velocity at the near wall region and overpredicts it at the center of the channel. This disagreement is possibly due to the wall treatment. The FFD used a simple no-slip wall boundary condition. This boundary treatment is proper for the laminar flow. However, the channel flow at $Re_\tau$ = 180 is turbulent (Kim et al. 1987). Therefore, in order to improve the accuracy, more advanced models for the wall are necessary.

Our previous work (Zuo and Chen 2007) found that the velocity profile predicted by the FFD did not satisfy the mass conservation. This investigation successfully solved this problem by fixing the pressure at a given point in the domain.

### Natural Convection Flow in a Tall Cavity

The airflow due to natural convection in a tall cavity is like that in a room with a heater in the winter. This study used a case with experimental data from Betts and Bokhari (1995). The cavity was 0.076 m wide and 2.18 m tall as shown in Figure 2. The right wall was heated at $T_2$ = 34.7 $^oC$ and the left wall cooled at $T_1$ = 15.1$^oC$. The corresponding Rayleigh number was 0.86 × $10^6$. The FFD simulation was carried out on 10 × 20 non-uniform grid cells (Figure 3) with a time step equal to 0.05 s.

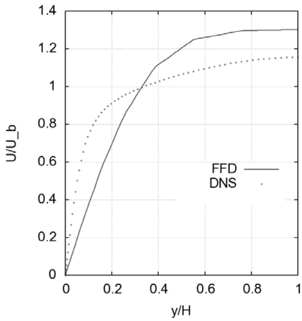



Figure 1. The comparison of mean streamwise velocity of the plane channel flow at $Re_{\tau}$ = 180, predicted by the FFD and DNS (Kim et al. 1987)

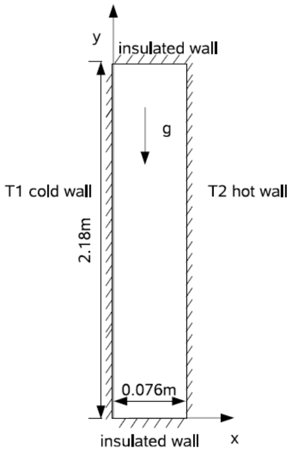



Figure 2. The sketch of natural convection in a tall cavity

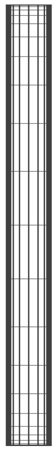
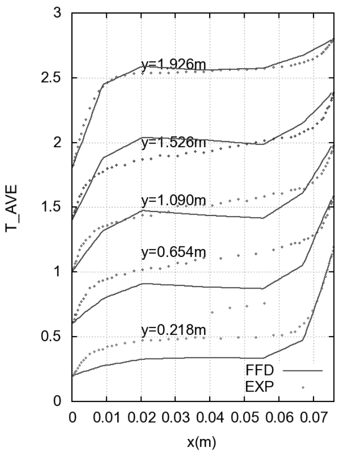
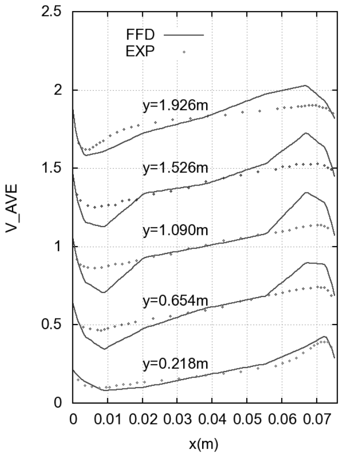


Figure 3. The mesh used in the case of the natural convection in a tall cavity

Figure 4 compares the predicted temperature and vertical velocity by the FFD with the corresponding experimental data. Although the temperature profiles predicted by the FFD are steeper at the near wall region and flatter at the center of the cavity, the agreement with the experimental data is acceptable considering the simple flow model used. The computed vertical velocities agree with the experimental data better at the center of the cavity than at the near wall regions. This is probably due to the overpredicted heat transfer from the walls by the FFD, which generated a larger buoyancy force and, consequently, a larger velocity near the walls.




(a) Air temperature




(b) Vertical air velocity

Figure 4. Comparison of the averaged air temperature and vertical air velocity predicted by the FFD with the experimental data (Betts and Bokhari 1995).

### Forced Convection Flow in a Room

The forced convection case used is based on Restivo's experiment (1979). Figure 4 shows the sketch of the experiment, where H was 3 m. The inlet height, $h_{in}$, was 0.168 m (0.056 H) and inlet velocity, $U_{in}$, was 0.455 m/s. The outlet height, $h_{out}$, was 0.48 m (0.16 H). Based on the inlet height and inlet velocity, the Reynolds number was 5000. Multiple boundary conditions, such as inflow, outflow and walls, were applied on the flow domain. The FFD used 36 × 36 non-uniform grid cells and a time step of 0.5 s (Figure 6).

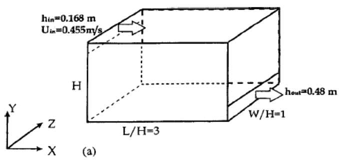



Figure 5. The sketch of a forced convection flow in a room

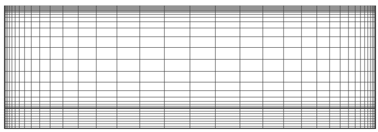


Figure 6. The mesh used in the case of forced convection flow in a room

Figure 7 compares the FFD results in two vertical and two horizontal lines across the room with the experimental data. The experimental data illustrates that the flow was complex because there was a secondary recirculation in the upper-right corner and another in the lower-left corner. The FFD can properly predict the velocity at the center of the room (x = H and 2H), but it did not work perfectly at the near wall regions (y = 0.028H and 0.972H). Two possible reasons may cause that problem. First, the grid resolution of the near wall region is coarse. Second, flow near the wall is very complex and current no-slip wall boundary condition is not proper. As discussed in channel flow section, to correctly capture the flow at the near wall region, one has to apply appropriate wall treatment.

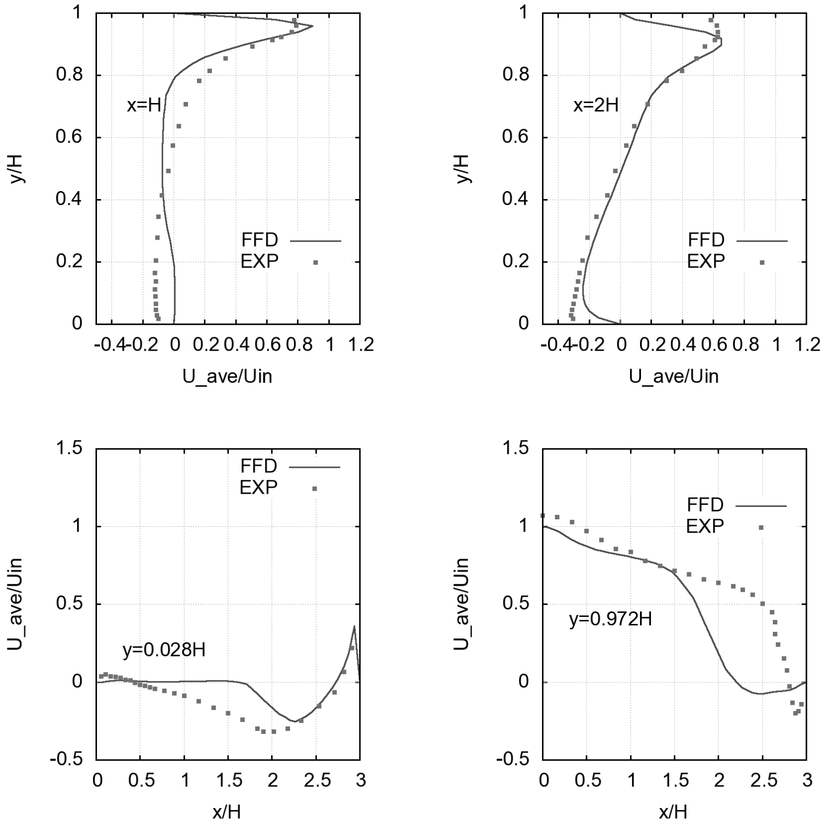



Figure 7. Comparison of horizontal air velocities by the FFD and the experimental data (Restivo 1979). The data are extracted at two vertical and horizontal sections across the room.

## DISCUSSION

This investigation evaluated also the computing speed of the FFD method. The evaluation defined a “speed enhancement” as $N = t_{physical} / t_{cpu}$, where $t_{cpu}$ is the elapsed CPU time used by the FFD and $t_{physical}$ the physical time of flow motion. Thus, real time simulation means $N = 1$. When $N > 1$, the FFD simulation is faster than real time.

For the three cases, the FFD simulations were faster than the real time on a HP workstation with an Intel Xeon (TM) CPU at 3.60 GHz. Table 1 lists the performance of the FFD simulations. The FFD ran much faster than real time in all the three cases.

However, the $N$ strongly depends on number of grids and time step size. For example, the forced convection case used finer grid (6.5 times), but even larger time step (10 times) than the natural convection case. Furthermore, the FFD did not solve temperature equation for the isothermal flow in the forced convection. Therefore, the FFD for the forced convection obtained more speed enhancement than the natural convection. Obviously, a coarse grid size and large time steps can accelerate the simulation but accordingly degrade the accuracy. Therefore, one has to find a trade-off between the computational performance and accuracy.

Table 1 Performance of the FFD simulations

| Case | Grids | Δt (s) | N |
|---|---|---|---|
| Channel flow | 64 × 32 | 0.1 | 6.1 |
| Natural convection | 10 × 20 | 0.05 | 25.4 |
| Forced convection | 36 × 36 | 0.5 | 98.6 |

## CONCLUSIONS

This paper introduced a scheme of fast fluid dynamics (FFD) method. The FFD has been used to compute airflow and temperature distributions for a fully developed plane channel flow, a natural convection flow in a tall cavity, and a forced convection flow in a ventilated room. The three flows represent the basic flow features in buildings. The corresponding experimental or DNS data from the literature for the three flows were used to compare the FFD results. The results show that the FFD can predict the airflows with acceptable accuracy at a speed 6 to 100 times faster than real time.

## NOMENCLATURE

| | |
|---|---|
| C | Contaminant concentration or air temperature |
| $K$ | Contaminant or thermal diffusivity |
| $f_i$ | Force |
| $P$ | Pressure |
| $S$ | Source |
| $\Delta t$ | Time step |
| $U_i$, $U_j$ | Velocity components in $x_i$ and $x_j$ directions, respectively |
| $x_i$, $x_j$ | Spatial coordinates |
| $\upsilon$ | Dynamics molecular viscosity |


## ACKNOWLEDGEMENTS

This project was funded by U.S. Federal Aviation Administration (FAA) Office of Aerospace Medicine through the Air Transportation Center of Excellence for Airliner Cabin Environment Research under Cooperative Agreement 04-C-ACE-PU. Although the FAA has sponsored this project, it neither endorses nor rejects the findings of this research. The presentation of this information is in the interest of invoking technical community comment on the results and conclusions of research.


## REFERENCES


1. P. L. Betts and I. H. Bokhari (1995) "New experiments on turbulent natural convection of air in a tall cavity", Proc. of IMechE Conference Transactions, 4th UK National Conference on Heat Transfer.
2. A. J. Chorin (1967) "A numerical method for solving incompressible viscous flow problems", Journal of Computational Physics, Vol. 2, 12-26.
3. R. Courant, E. Isaacson and M. Rees (1952) "On the solution of nonlinear hyperbolic differential equations by finite differences", Communication on Pure and Applied Mathematics Vol. 5, 243–255.
4. J. Kim, P. Moin and R. Moser (1987) "Turbulence statistics in fully-developed channel flow at low Reynolds-number", Journal of Fluid Mechanics, Vol. 177, 133-166.
5. A. Robert, C. Turnbull and J. Henderso (1972) "Implicit time integration scheme for baroclinic models of atmosphere", Monthly Weather Review, Vol. 100, 329-335.

6. J. Stam (1999) "Stable fluids", Proc. of SIGGRAPH'99.
7. L. Wang (2007) "Coupling of multizone and CFD programs for building airflow and contaminant transport simulations", Ph.D. Thesis, Purdue University.
8. W. Zuo and Q. Chen (2007) "Validation of fast fluid dynamics for room airflow", Proc. of International Symposium on Heating, Ventilating and Air Conditioning, Beijing, China.