Zuo, W. and Chen, Q. 2009. "Real time or faster-than-real-time simulation of airflow in buildings," *Indoor Air,* 19(1), 33-44.

Real time or faster-than-real-time simulation of airflow in buildings

Wangda Zuo **Qingyan Chen***

National Air Transportation Center of Excellence for Research in the Intermodal Transport Environment (RITE)
School of Mechanical Engineering, Purdue University, West Lafayette, IN 47907-2088

Abstract

Real time flow simulation is crucial for emergency management in buildings, such as fire and accidental or intentional release of chemical/biological agents (contaminants). The simulation results can then be used to impose proper measures to minimize casualties. The computational fluid dynamics (CFD) is accurate, but too time consuming. Nodal models are fast, but not informative. To obtain a fast and informative solution, this study proposes an intermediate approach between nodal models and CFD by introducing a Fast Fluid Dynamics (FFD) method. This investigation used the FFD methods with and without turbulence treatments to study systematically four basic flows in buildings and compared the numerical results with the corresponding CFD results and the data from the literature. The results show that, on one side, the FFD can offer much richer flow information than nodal models do, but less accurate results than CFD does. On the other side, the FFD is 50 times faster than the CFD. The results also show that the FFD with the laminar assumption has the best overall performance on both accuracy and speed. It is possible to conduct faster-than-real-time flow simulations with detailed flow information by using the FFD method.

Key Words: Fast Fluid Dynamics; Computational Fluid Dynamics; Contaminant Dispersion; Air Distribution; Real Time

Practical Implication

The paper introduces a Fast Fluid Dynamics (FFD) method, which can simulate airflow and contaminant dispersion in buildings with real time or fast-than-real-time speed and provide informative solutions. As an intermediate approach between nodal models and the CFD, the FFD can be a very useful tool for emergency management in case of fire and accidental or intentional release of chemical or biological agents in a building or around the buildings. The FFD can also be used as a preliminary test tool for fast assessment of indoor airflows before a detailed CFD analysis.

*Corresponding author. Tel.: +765-496-7562, Email address: yanchen@purdue.edu

Nomenclature

C	Scalar variables, such as smoke or contaminant concentrations
d	Particle diameter
f_i	Force
k	Contaminant or thermal diffusivity
L	Length scale
N	Ratio of real time of flow motion over elapsed time used by simulations
P	Pressure
Re	Reynolds number
Re_m	Reynolds number based on bulk mean velocity
S	Source
St	Stokes number
T	Temperature
$t_{physical}$	Physical time of flow motion
$t_{elasped}$	Elapsed time used by simulations
U	Horizontal velocity; particle velocity
U_i, U_j	Velocity components in x_i and x_j directions, respectively
U_m	Bulk mean velocity
V	Vertical velocity
x_i, x_j	Spatial coordinates
y^+	Spatial coordinates in wall units
Δt	Time step
μ	Molecular viscosity
ν	Dynamics molecular viscosity
ν_t	Turbulent dynamic viscosity
ρ_p	Density of contaminant particles

Background

According to the United States Fire Administration (USFA, 2007), 3,245 civilians and 106 firefighters lost their lives in fires, with an additional 16,400 civilians injured as the result of fire in 2006. Smoke inhalation is responsible for most of fire-related injuries and deaths. Meanwhile, accidental release of chemical/biological agents in buildings also happens occasionally, such as the accidental nerve gas release in the United National building in New York on August 30, 2007. Furthermore, chemical/biological warfare agents can also be used by terrorists to attack civilians in enclosed environments. Examples are the attack with nerve agent sarin at Tokyo subway in 1995 and the discovery of the biological toxin ricin in the Senate Office Building in Washington, D.C. in 2004.

Faster-than-real-time prediction of the smoke or contaminant (chemical/biological agent) transport can provide information how the smoke or contaminant is transported or dispersed in buildings. If the prediction is accurate and informative, emergency management personnel can use the prediction to impose proper measures to evacuate the occupants in the buildings to minimize casualties. Unfortunately, none of current modeling technologies can meet such requirements. Either their computing speed is too slow or their results are not informative.

For example, Computational Fluid Dynamics (CFD) is widely applied in the simulation of smoke and contaminant transport in enclosed environments (Ferziger and Peric, 2002). CFD can provide detailed and accurate information about the distributions of air velocity, temperature, and smoke/contaminant concentrations (Chen et al., 2007, Nielsen, 2004, Zhai et al., 2007). Unfortunately, a CFD simulation for 10-minute transient smoke/contaminant transport in a single enclosed space, such as a hotel lobby, would take a day of computing time on a PC. Obviously, this method is only valuable for aftermath analysis but too slow for imposing evacuation strategies on real-time.

On the other hand, by representing the air properties and contaminant concentrations in an enclose space with only one or several nodes, the nodal models, such as multizone models (Axley, 2007) and zonal models (Megri and Haghighat, 2007), may provide faster-than-real-time information of transient smoke/contaminant transport in a building. However, the nodal models may not provide detailed information of smoke/contaminant transport for emergency management (Wang and Chen, 2008). Their predictions are also not accurate when the flow is with strong effect of momentum, buoyancy, and contaminant gradient, especially under transient conditions (Schaelin et al., 1994, Wang and Chen, 2007).

Forgoing discussion shows that current modeling techniques are difficult to provide informative and real-time smoke/contaminant transport information for a building at the same time. Therefore, an intermediate method is necessary to fill the gap between nodal models and CFD. For instance, one can seek methods that have been successfully applied in related fields, such as flow simulation for computer games, which could be used for buildings. Stam (1999) developed a Fast Fluid Dynamics (FFD) method for computer games that recreates a plausible real-time flow motion based on the Navier-Stokes (NS) equations. His scheme is so efficient that a single PC can support real time interactions between players and the computer games. Using the FFD scheme, Fedkiw et al. (2001) simulated smoke motions; Harris (2003) calculated cloud movements; and Liu et al (2004) computed the flow around complex obstacles. All were realized in real-time or faster-than-real-time.

The successful applications of the FFD method in computer games indicate a great potential for real-time simulation of airflow and contaminant transport in buildings. However, current FFD applications in game industry and computer visualization are satisfactory with plausible solutions and the accuracy of the FFD simulation is yet strictly verified. Meanwhile, no information was provided on the speed of the FFD simulations. Therefore, to introduce the FFD into building simulation, it is essential to evaluate the accuracy and speed of the FFD method and to compare them with current methods, such as CFD. This paper reports our effort on validating the FFD for various airflows in buildings with the well-known experimental data from the literature and determining the computing speed of the FFD method.

Research Approach

The FFD method developed by Stam (1999) solves the advection term of NS equations with a first order semi-Lagrangian scheme (Courant et al., 1952), computes the diffusion term with an implicit Gauss-Seidel iteration, and decouples pressure and velocity with a pressure-correction projection scheme (Chorin, 1967). With these efforts, one can achieve real-time flow simulation. Namely, the FFD solves the following continuity and NS equations for time-dependent incompressible fluid:

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

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

where U_i and U_j are fluid velocity components in x_i and x_j directions, respectively; ν is kinematic molecular viscosity; P is pressure; and f_i are forces, such as buoyancy force and other external forces. This study assumed that the contaminants in buildings are gaseous or particles with very small Stokes numbers, which is defined by

$$St = \frac{\rho_P d^2 U}{18 \mu L} , \tag{3}$$

where ρ_p is the particle density, d is the particle diameter, μ is the molecular viscosity of air, U is the characteristic flow velocity, and L is a length scale. This assumption allows us to approximate motions of small particles to follow the air paths (Crowe et al., 1996). The contaminant concentrations can then be analogized to a scalar variable, namely temperature in this study. Therefore, one would solve the contaminant concentrations by using the Eulerian approach. Note that other approaches, such as Lagrangian method, may be necessary if the Stokes number of particles is large. 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 represents contaminant concentration or air temperature; k is contaminant or thermal diffusivity; and S is a source. In each time step, the FFD solves the NS equations (2) in four stages:

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

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 Δt is the time step. The second stage is to solve the diffusion term through a first order implicit scheme by Gauss-Seidel iteration:

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

By applying the implicit scheme, the simulation is always stable even when the CFL number is much larger than one. The CFL number is defined by $CFL = U_i \, \Delta t/\Delta x$, where Δx is mesh size (Courant et al., 1928). The third stage is to solve the advection term:

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

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

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

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) with equation (1). The projection scheme first solves a Poisson equation for pressure:

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

Then the scheme ensures the conservation of mass by correcting the velocities as

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

where $U_i^{(4)}$ is the velocity satisfying the continuity equation (1).

The above part shows that although the FFD solves the NS equation as the CFD does, it is different from the CFD. Since the computing speed is the most important issue in the FFD simulation, the FFD uses simple and low order schemes to reduce the computing cost. For example, it uses the linear interpolation in the semi-Lagrangian approach instead of a high order non-linear interpolation. The pressure-correction projection method used by the FFD is also the simplest one among the different projection schemes. The FFD also applies low order discretization scheme (1st order for time and 2nd order for space) and this can generate too much numerical dissipation (Fedkiw et al., 2001). As a result, the FFD has a lower computing cost but lower accuracy than the CFD does.

Since most indoor airflows are turbulent, this study also implemented two different turbulence treatments into the FFD in addition to the Stam's laminar assumption. One used a constant turbulent viscosity, ν_t, to be 100ν; and the other applied a zero-equation turbulence model (Chen and Xu, 1998). This investigation implemented the FFD method by creating a program using C-programming language. This study also employed a commercial CFD program FLUENT (http://www.fluent.com) for comparison of accuracy and computing speed between the FFD and CFD.

This study assessed the accuracy and speed of the FFD method by comparing the FFD results with the CFD ones. Table 1 shows the comparing matrix. The laminar flow comparison was used to assess the conventional FFD numerical scheme. The zero-equation was added to appreciate the impact of turbulence. Since the FFD and CFD used different numerical schemes, the performance of turbulence models could be different. The FFD approach using $\nu_t = 100\nu$ was a simple approach to solve turbulent flow and had limited success in results shown in the literature. Thus, it was also selected for evaluation. The RNG k-ε model is the most popular one for CFD at present. By adding it to the comparison would give us good indication of the performance of the different approaches.

Table 1. Different approaches used to assess the FFD method

FFD method	*CFD method*
Laminar flow	Laminar flow
Turbulent flow with zero-equation model	Turbulent flow with zero-equation model
Turbulent flow with $\nu_t = 100\nu$	Turbulent flow with RNG k-ε model

Results

This investigation selected four typical indoor airflows as test cases, which represent the most basic elements of flows found in buildings: (1) a fully developed turbulent flow in a plane channel (Kim et al., 1987); (2) a forced convection flow in a ventilated room (Restivo, 1979); (3) a natural convection flow in a tall cavity (Betts and Bokhari, 2000); and (4) a mixed convection flow in a ventilated room (Blay et al., 1992). Those cases are with high quality flow and temperature data but unfortunately without contaminant concentration data. As discussed previously, the air temperature distribution can be analogized to contaminant concentration if the contaminants are gases or particles with small Stokes number. One can expect the same accuracy for contaminant concentrations from the results of air temperature.

In the reality of emergency management, the dispersion of smoke or contaminants is transient. Hence the FFD and CFD should perform transient simulations although the flows for the four cases were steady. The FFD simulations show that the flows are slightly oscillating even they are steady state. This is the nature of turbulent flows and is consistent with the observation in experiments. To compare with the corresponding data from the literature, the results of the FFD and CFD simulations presented in this paper were averaged over time.

To make the computing speed compatible, the number of iterations at each time step for both the FFD and CFD simulations was fixed. The two programs used the same grid number and time step size for the same case. The grid distributions for FFD can be different from those for CFD simulations due to the requirements of the wall functions for turbulent flows. But this should not have an impact on the computing speed.

The FFD and CFD results for the four selected cases with different approaches shown on Table 1 are presented in this section. The corresponding data for the four cases in the literature are used as benchmarks for assessing the accuracy. Since this study emphasized on fast simulation, the grid resolutions were coarser than those for typical CFD simulations nowadays but comparable to those used in the 1980s and 1990s. To fully appreciate the accuracy of CFD simulations for indoor airflows, one can refer to the recent results from Zhang et al. (2007).

Fully Developed Turbulent 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 turbulent flow in a plane channel as a test case. Based on mean bulk velocity, U_m, and the channel half-width, H, the flow Reynolds number studied was 2800, which could create turbulence in the flow. Kim et al. (1987) did direct numerical simulation (DNS) for this flow and their data were used as a reference. This study applied a simple uniform velocity inlet condition for both the FFD and CFD simulations.

All the simulations were performed on 64 × 32 non-uniform grids and with a time step of 1.0s. Figure 1 compares the normalized mean streamwise velocity obtained by the FFD and CFD approaches with the DNS data (Kim et al., 1987). Figure 1 shows that the CFD with the laminar assumption predicted the velocity profile well because the flow was not highly turbulent. The prediction of the FFD with the laminar assumption was not as good as that of the laminar CFD, perhaps this is due to the inefficient and low order schemes used by the FFD. The results demonstrate the differences between the CFD and the FFD solvers. The FFD prediction with $\nu_t = 100\nu$ was much worse. Clearly, the turbulent level was well over-estimated. The CFD with the RNG k-ε model led to the best result. The prediction of CFD with the zero-equation model was also close to the DNS data. The FFD with the zero-equation model did not work as well as the CFD. The reason could again be attributed to the solver used. However, the predicted profile was better than the one by laminar FFD.

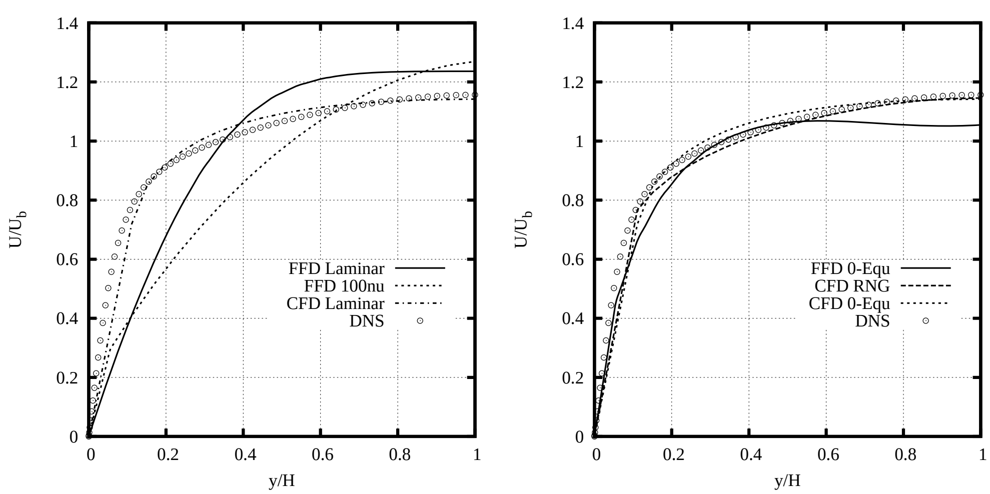


Fig. 1 The comparison of normalized streamwise velocity of the plane channel flow predicted by the FFD and CFD with the DNS data

Forced Convection Flow in a Room

The forced convection case is based on Restivo's experiment (Restivo, 1979). Figure 2 shows the sketch of the experiment, where $L = 3H$. The inlet height, h_{in}, was 0.056 H and the outlet height, h_{out}, was 0.16 H. The Reynolds number was 5000 based on the inlet height and inlet velocity, which can lead to turbulent flow in the room. The experiment was designed to produce two-dimensional flow field. This study employed a 36 × 36 non-uniform grid and and a time step of 0.5s for both the FFD and CFD simulations.

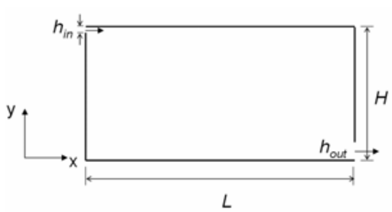
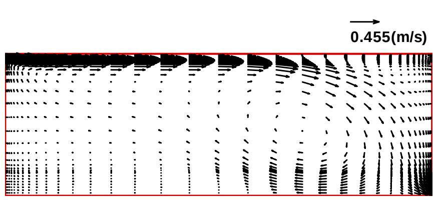


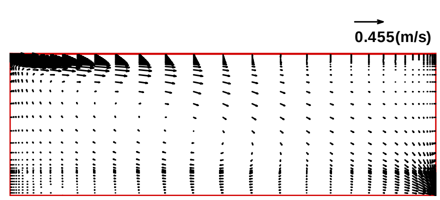
Fig. 2 The sketch of the forced convection flow in a room

Figure 3 compares the velocity fields and Figure 4 the velocity profiles in two vertical sections predicted by the FFD and CFD approaches. The experimental data from the literature on the two sections are also used for comparison. The computed velocity field by the CFD with the RNG k-ε model (Figure 3f) is the same as that by Chen (1995). The corresponding velocity profiles also agree with the experimental data. The results predicted by the CFD with the zero-equation model and with laminar assumption are close to those by the RNG k-ε model and experimental data, except the eddy size in the upper-right and bottom-left corners. The performance of the zero-equation model is better than that of the laminar assumption in the CFD simulations. The differences in the results are due to the turbulence models used.

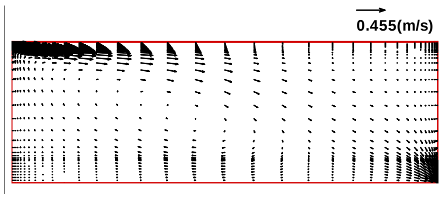
The FFD with the laminar assumption did a reasonable job. The whole velocity distribution by the FFD laminar solver (Figure 3a) seems not as good as the one by the CFD laminar solver. But it computed the velocity profiles at the two specific locations closer to the experimental data than the CFD laminar solver did (Figure 4). The FFD with ν_t = 100 ν (Figure 3b) and with the zero-equation model (Figure 3c) could not predict the big recirculation well. The possible reason is that both turbulence treatments over-estimated the turbulence viscosity.



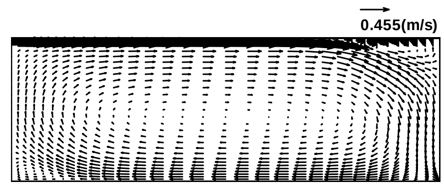
(a) FFD with laminar assumption



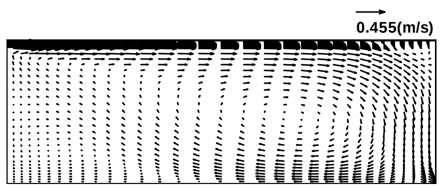
(b) FFD with ν_t=100ν



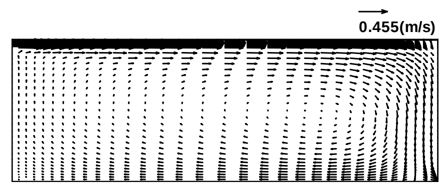
(c) FFD with zero-equation model



(d) CFD with laminar assumption

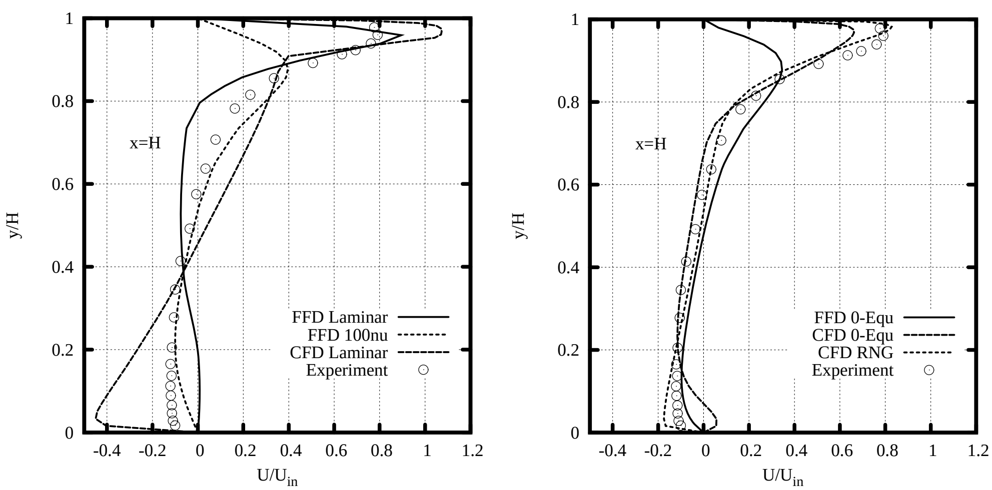


(e) CFD with zero equation model

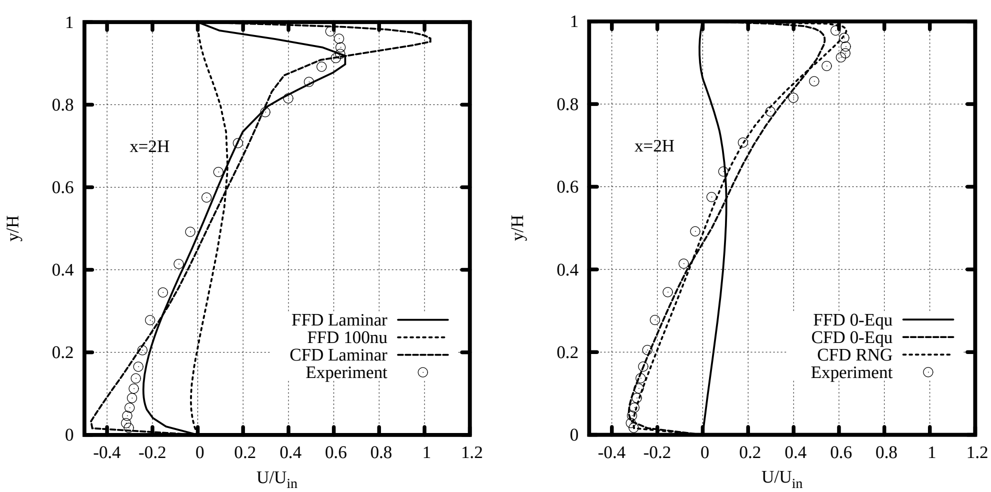


(f) CFD with RNG k-ε model

Fig. 3 Comparison of velocity field predicted by the FFD and CFD



(a) U at x = H



(b) U at x = 2H

Fig. 4 Comparison of horizontal air velocities predicted by the FFD and CFD with the experimental data

The FFD can also simulate contaminant transportation as mentioned previously. Our FFD code has an interactive interface that allows releasing a contaminant in any location of the simulated domain by simply clicking the mouse. Then the code will calculate the transport of the contaminant and visualize it on screen. Figure 5 shows a screen print of contaminant (white smoke) dispersion for this case. One can clearly see turbulent vortices of smoke in the center of the room. A part of the smoke is expelled from the outlet at the bottom-right corner. The distribution of the smoke looks plausible. Unfortunately, the experimental data of contaminant concentration is not available for this case so that the accuracy can not be validated yet.

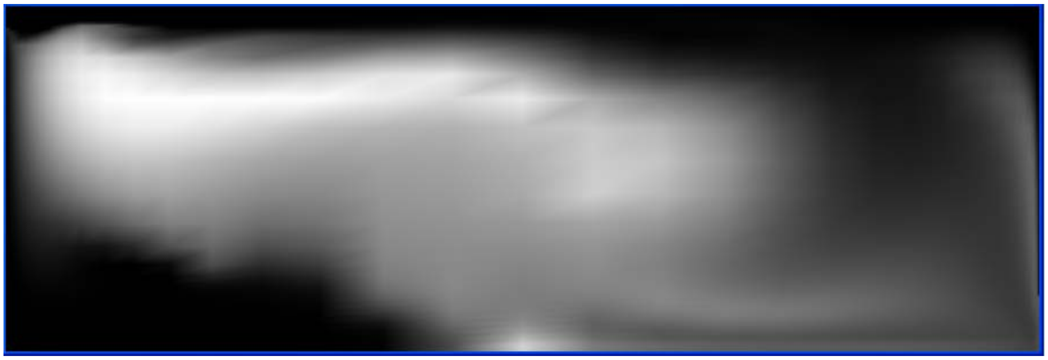


Fig. 5 A screen print of contaminat dispersion simulated by FFD

Natural Convection Flow in a Tall Cavity

The airflow due to natural convection in a tall cavity is a typical window problem and has been classified as a basic flow feature in buildings. This study used a case with the experimental data from Betts and Bokhari (2000). The cavity was 0.076 m wide and 2.18 m tall as shown in Figure 6. The left wall was cooled at T_1 = 15.1°C and the right wall heated at T_2 = 34.7°C. The corresponding Rayleigh number was 0.86×10^6, which can create a turbulent flow (Betts and Bokhari, 2000). The experiment was set up to have two-dimensional velocity and temperature distributions. The FFD and CFD simulations were performed with a 10 × 20 non-uniform grid distribution and a time step of 0.05s.

Figure 7 compares the profiles of air temperature and vertical velocity predicted by the FFD and CFD approaches with the experimental data. Since this study did not apply the optimized grid resolution and distribution for the turbulence models, the CFD predictions in this study are not as good as those by Zhang et al (2007). The results obtained from this investigation show that the CFD with the RNG k-ε model worked well for velocity but rather poor for the air temperature. Clearly, the problem could be attributed to the grid distribution and resolution. The CFD with the zero-equation model computed properly the velocity profiles but not the temperature profiles. But the prediction by the CFD with the zero-equation model is better than that by the FFD with the zero-equation model. The recommended turbulent Prandtl number for the zero-equation model in CFD simulation is not the optimized one for the FFD. One can improve the FFD prediction by carefully tuning the turbulent Prandtl number. However, that is not the focus of this paper.

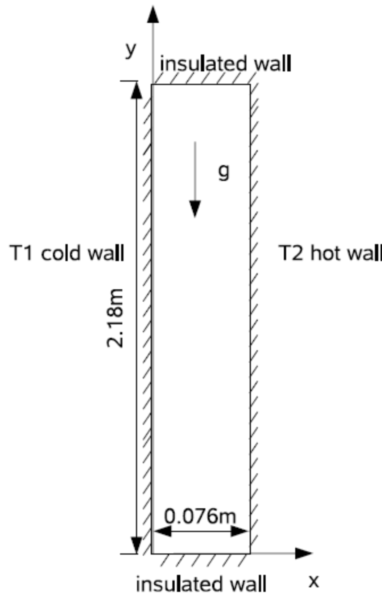
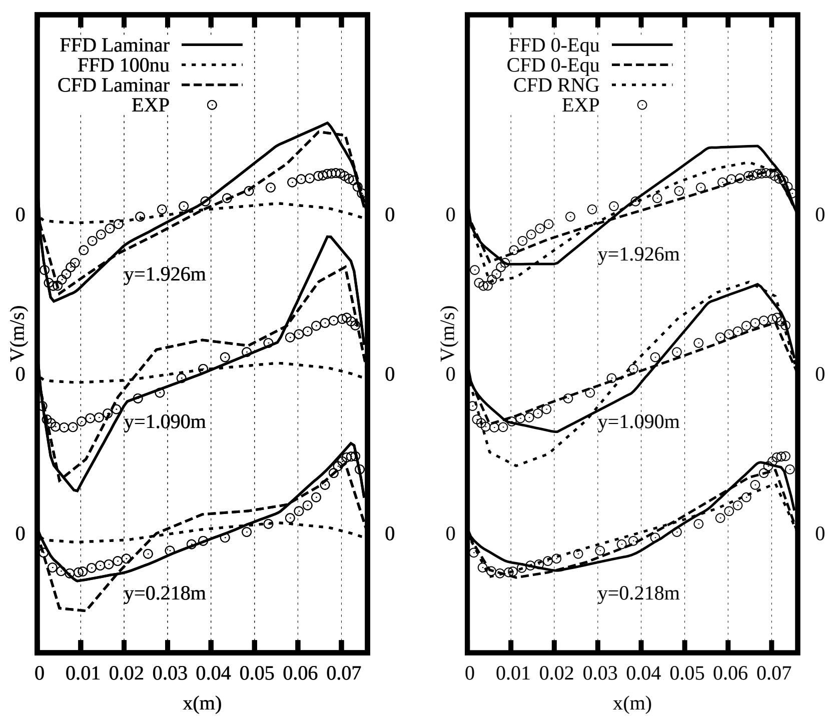
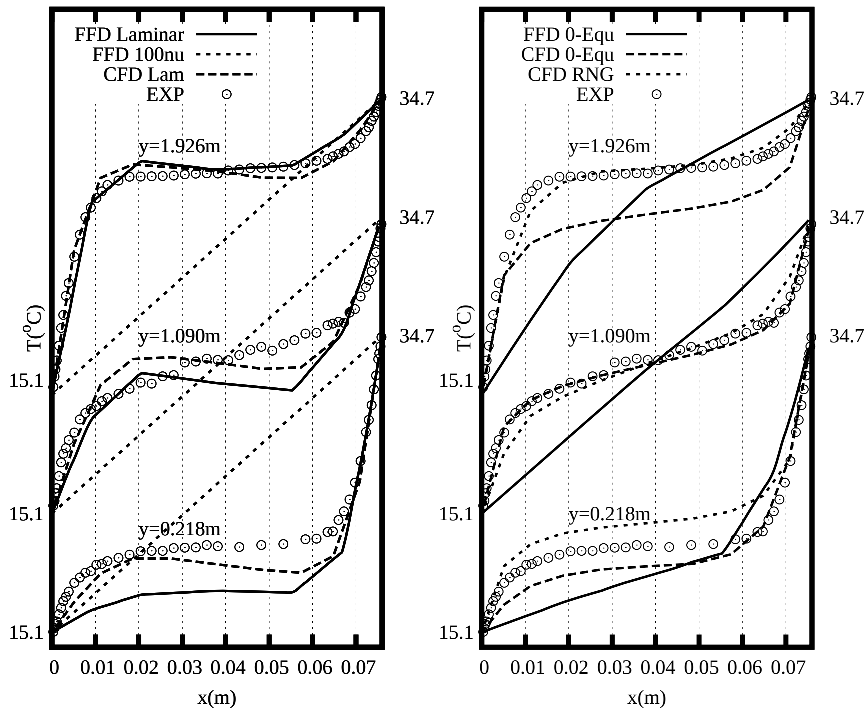


Fig. 6 The sketch of natural convection in a tall cavity

As expected, the CFD with the laminar assumption did a much poorer job than those with turbulence models. The FFD laminar solver did a better job than the one with turbulent models. The FFD simulation with ν_t = 100 ν failed to correctly calculate the trend of the air temperature and velocity profiles. One possible reason might be the flow with such a low Rayleigh number is at quite low turbulent level and the turbulent models over-predicted the turbulence. The other possible reason is the recommended turbulent Prandtl number for the CFD simulation is not good for the FFD.



(a) Vertical velocity profiles



(b) Temperature profiles

Fig. 7 Comparison of air temperature and vertical velocity profiles predicted by the FFD and CFD with the experimental data

Mixed Convection Flow in a Room

The mixed convection case is based on the experiment conducted by Blay et al. (1992). Figure 8 shows the sketch of the experiment, where *L* was 1.04 m. The inlet height and outlet height were 0.018 m and 0.024 m, respectively. The inlet air velocity and temperature were 0.455 m/s and 15°C, respectively. The temperature of the upper, left, and right walls, T_l, was 15°C; and the temperature of the lower wall, T_h, 35°C. The flow was driven by both the inertia and buoyancy forces. The experiment was designed to have a two-dimensional airflow pattern. Our simulations used a 20 × 20 non-uniform grid distribution with a time step of 0.02 s.

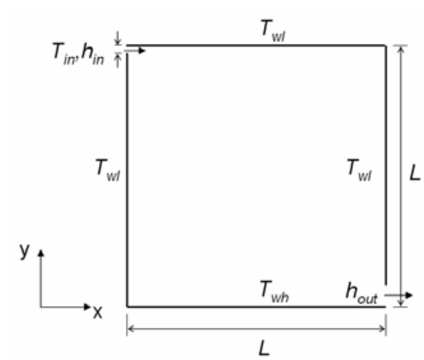
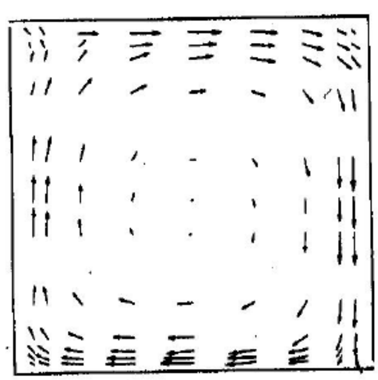


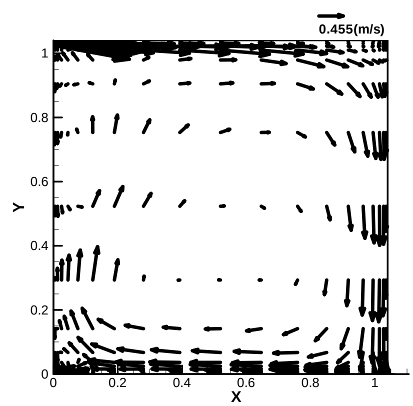
Fig. 8 The sketch of the mixed convection flow in a room

Compared with the experimental results in Figure 9(a), the FFD with the laminar assumption (Figure 9b) correctly computed the large clockwise recirculation in the center of the model room. The shape of recirculation bubble is similar to the measured one. Interesting enough, the FFD laminar solver also predicted the small secondary recirculations at the upper-right, lower-left, and upper-left corners, although they are larger than the measured ones. The FFD simulation with ν_t = 100 ν (Figure 9c) predicted the recirculation in a wrong opposite direction. This is because it over-calculated the buoyancy force. The FFD with the zero-equation model correctly computed recirculation direction but it incorrectly generated another big recirculation at the left side of the room (Figure 9d). The possible reason for the failure of turbulence models in the FFD simulation is that the turbulent models are not designed and optimized for the FFD.

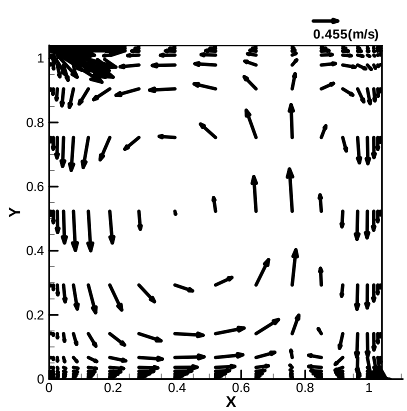
All the CFD approaches also correctly calculated the clockwise recirculation in the center of the room. The predictions by the CFD simulations with the zero-equation model and with the RNG k-ε model are in good agreement with the experimental data. However, the laminar CFD solver computed a poor shape of the recirculation bubble due to the lack of turbulence treatment.



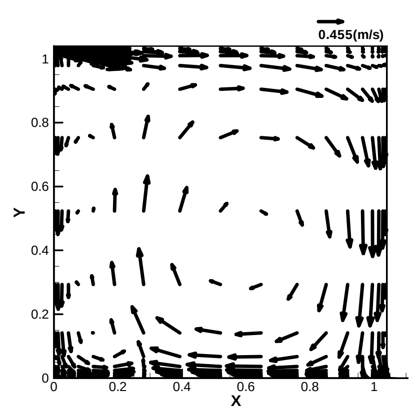
(a) Experiment (Blay et al., 1992)



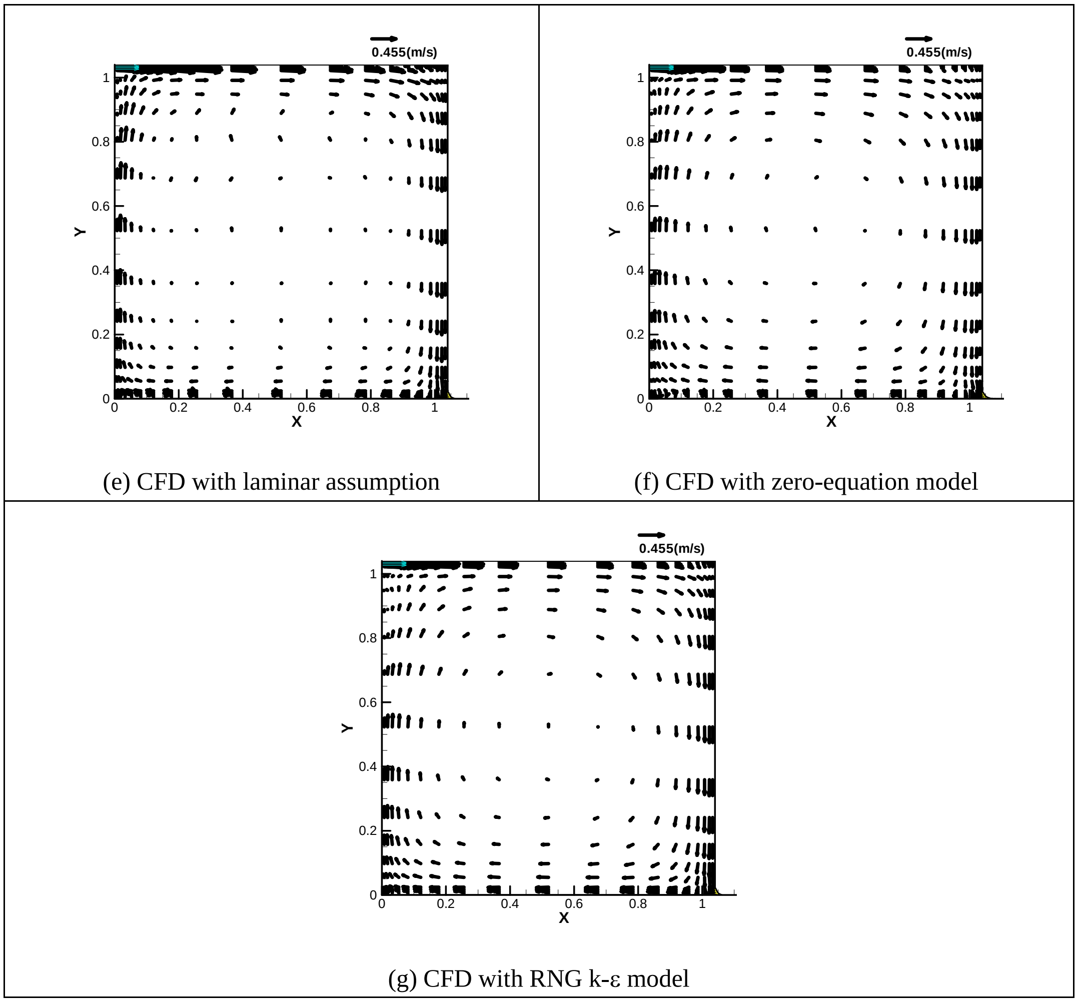
(b) FFD with laminar assumption



(c) FFD with ν_t = 100 ν



(d) FFD with zero-equation model



(e) CFD with laminar assumption

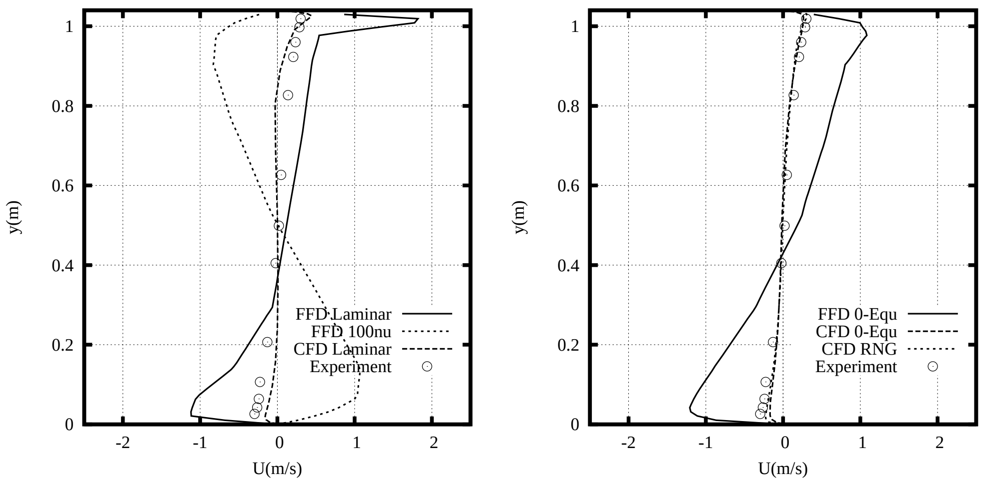
(f) CFD with zero-equation model

(g) CFD with RNG k-ε model

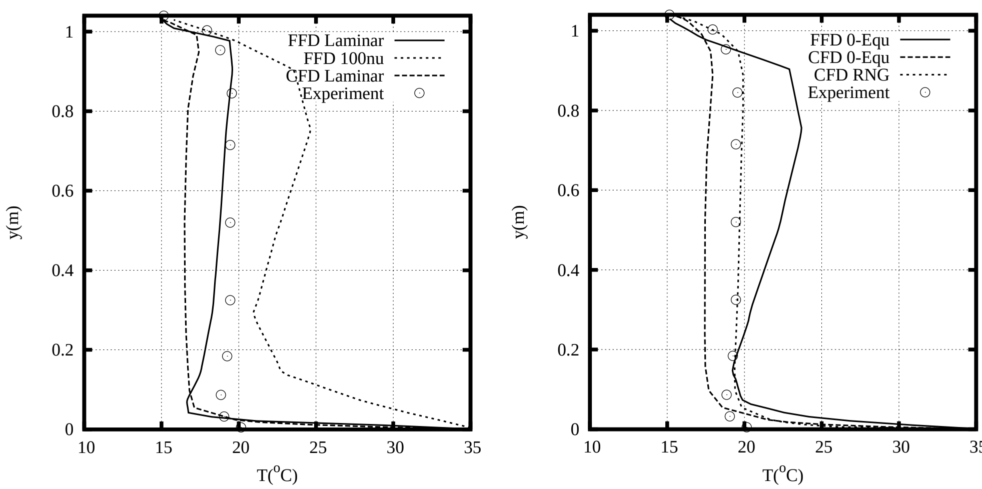
Fig. 9 Comparison of the velocity field of the mixed convection flow in the model room by the FFD and CFD with the experimental data.

Figure 10 compares the FFD and CFD predictions in the mid-width section of the model room with the experimental data. The FFD with the laminar assumption over-predicted the near wall velocities and slightly under-predicted the air temperature in the room. Compared to the CFD laminar solver, the FFD laminar solver produced a worse velocity profile but a better temperature profile.

The FFD simulation with $\nu_t = 100\ \nu$ predicted a wrong solution. The FFD simulation with the zero-equation model provided much worse results than the FFD with the laminar assumption. Both the CFD simulations with the RNG k-ε model and with the zero-equation model generated good velocity profiles and the former also properly calculated the temperature profile. But the CFD with the zero-equation model generated a lower temperature distribution in the room. It seems that the zero-equation model did not calculate correctly the heat transfer from the walls. The differences between the FFD and CFD are again due to the different solvers used.



(a) U at x = 0.5 L



(b) T at x = 0.5 L

Fig. 10 Comparison of air velocity and temperature profiles predicted by the FFD and CFD with the experimental data

Discussion

Accuracy

By combining all the results in the previous section, the FFD simulations can capture major features of the flow fields, such as flow directions and large recirculations. However, the FFD is not as accurate as the CFD.

As expected, the use of a turbulence model can significantly improve the CFD simulations. The RNG k-ε model is the best and the laminar assumption is the worst for the four cases tested. However, the use of a turbulence model may not improve the performance of the FFD. In fact, the constant viscosity assumption has made the performance of the FFD worse. The FFD with the laminar assumption has the best overall accuracy among the three FFD approaches. The reason is that current turbulence models are developed for CFD applications and they may not be suitable for the FFD because the FFD uses a different solver. If turbulence models are going to be applied in the future for the FFD, they must be further developed to work with the solvers.

The results of the FFD simulations are not as good as those of the CFD simulations with the same treatment for turbulence. The major reason is that the FFD solver is much simpler than the CFD one. The simplifications used in the solver seem relax the accuracy although they reduce the computing time.

It is a pity that none of the four cases selected have experimental data of transport and dispersion of gaseous contaminants. Thus, a validation on the contaminant transport is yet completed. Since the governing equation for the gaseous contaminant transport is very similar to that for air temperature, one can imagine the performance of the FFD for contaminant transport predictions.

It should be note that the effort of this investigation is not to seek a method to replace the CFD but to find a compromise between the CFD and zonal models. Although the accuracy of the FFD approaches is not as good as that of CFD, it is much more informative than the well-mixed assumptions typically used in multizone models.

Speed

Another objective of this study was to make use of the fast speed of the FFD method. It is thus important to compare the computing speed of the FFD with that of the CFD. This investigation defined a computing time ratio, N, as

$$N = t_{physical} / t_{elapsed} \qquad (11)$$

where $t_{physical}$ is the physical time of flow motion and $t_{elapsed}$ is elapsed computing time used by the FFD or CFD simulations. When $N = 1$, the simulation is real time; and when $N > 1$, the simulation becomes faster than real time. The N is therefore a good indicator for evaluating the computing speed for a flow simulation.

All the four cases simulated by the FFD and CFD were carried on a HP workstation with a single Intel Xeon (TM) CPU at 3.60 GHz. Table 1 lists the computing time ratio of the FFD and CFD simulations. Note that the FFD and CFD used the same grid number and time steps for the same case. Clearly, all the FFD simulations are faster than real time. The FFD simulations with the laminar assumption and with $\nu_t = 100\ \nu$ are the fastest. The FFD simulations with $\nu_t = 100\ \nu$ are with the same speed of the FFD laminar solver since it needs no additional computing effort. The FFD simulations with the zero-equation model are 23%-41% slower than that with laminar assumption. The CFD simulations with the laminar assumption and with the zero-equation model are faster than the one with the RNG k-ε model. But all the CFD approaches are around 50 times slower than the FFD approaches.

Table 2 Comparison of computing speed of the FFD and CFD simulations

Case	Channel flow	Natural	Forced	Mixed
Grids	64 × 32	10 × 20	36 × 36	20 × 20
Δt (s)	1.0	0.05	0.5	0.02
$N_{FFD,Laminar}$	72.3	25.4	49.9	3.91
$N_{FFD,100\upsilon}$	72.1	25.7	48.2	3.89
$N_{FFD,0\text{-}Equ}$	42.4	15.8	32.3	2.69
$N_{CFD,Laminar}$	1.76	0.27	1.37	0.079
$N_{CFD,0\text{-}Equ}$	1.68	0.27	1.35	0.075
$N_{\mathrm{CFD,RNG}}$	0.96	0.16	0.88	0.045

The N strongly depends on the number of grids, time step size, and flow conditions. Our experience shows that the time step size is flow dependent for the same accuracy. The grid number is of course related to geometry and the flow complexity. All the examples shown here used as fewer grids and as larger time steps as possible. Reducing the grid number and increasing the time step further may make the results unacceptable in terms of accuracy for emergency management. Therefore, there is trade-off between the computing speed and accuracy. In addition, many other factors could also influence the speed and accuracy of a flow simulation, such as the order of implicit timing scheme, application of the multigrid method, and numerical solver for the Poisson equation.

The computing performance reported here was obtained with two-dimensional grid distributions. The computing speed of the FFD by using three-dimensional grids is 40%-50% slower than that by using the same amount of two-dimensional grids. This reduction on the computing speed is reasonable because the three-dimensional computations used more complex formula.

It is possible to further accelerate the FFD simulations by running it on computer cluster with multiple processors or on Graphic Processing Units (GPU). GPUs achieve high performance through parallelism, because they are capable of processing simultaneously multiple vertices and pixels. Hence, a single GPU can work like multi-CPUs paralleled. Meanwhile, recent rapid development in GPU programmability makes it possible to use GPUs for general purpose computation, such as flow simulations demonstrated by (Harris, 2003, Scheidegger et al., 2005). Liu et al. (2004) showed that GPU was 14 times faster than CPU for the FFD simulations. Furthermore, GPU performance has increased at a much faster rate than CPUs and the trend will continue in the future as shown in Figure 11. Therefore, to develop a FFD program for GPU computation would make real time simulation possible for airflow and contaminant transport in a building with reasonable amount of grids and sufficient fine time steps.

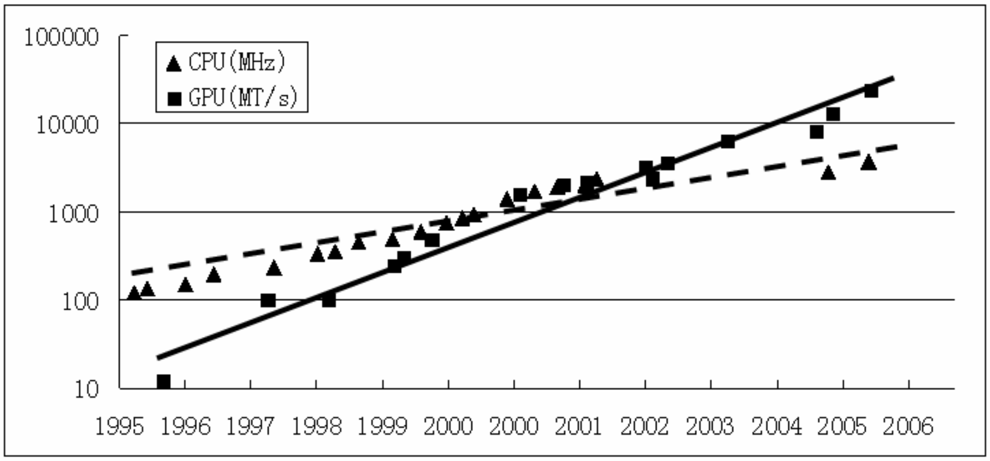


Fig. 11 The hardware performance increasing of GPU (NAVIDA) and CPU (INTEL) over the past ten years

Conclusions

This paper introduced a scheme of Fast Fluid Dynamics (FFD) method. The FFD approaches with and without turbulence models have been used to compute airflow and temperature distributions for a fully developed plane channel flow, a natural convection flow in a tall cavity, a forced convection flow in a ventilated room, and a mixed convection flow in a ventilated model room. The four flows represent the basic flow features in buildings. To compare the FFD performance, CFD simulations were also conducted for the comparison. The results showed that the FFD methods can predict the airflow patterns at a speed 4 to 100 times faster than real time with 200 to 2000 two-dimensional grids. The FFD simulations without using a turbulence model performed better than those with turbulence models. The FFD simulations were around 50 times faster than the CFD simulations. Although the accuracy of the FFD results is not as good as those of CFD, the FFD can offer much richer information than the nodal models. The FFD can be a very useful intermediate method between nodal models and CFD for fast simulation of airflow and contaminant dispersion in a building and around buildings.

Acknowledgements

This project is funded by the U.S. Federal Aviation Administration (FAA) Office of Aerospace Medicine through the National Air Transportation Center of Excellence for Research in the Intermodal Transport Environment under Cooperative Agreement 07-C-RITE-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 the research.

References

Axley, J. (2007) "Multizone airflow modeling in buildings: History and theory", *HVAC&R Research,* **13**, 907-928.

Betts, P.L. and Bokhari, I.H. (2000) "Experiments on turbulent natural convection in an enclosed tall cavity", *International Journal of Heat and Fluid Flow,* **21**, 675-683.

Blay, D., Mergui, S., and Niculae, C. (1992) "Confined turbulent mixed convection in the presence of horizontal buoyant wall jet", *Fundamentals of Mixed Convection,* **213**, 65-72

Chen, Q. (1995) "Comparison of different k-ε models for indoor air-flow computations", *Numerical Heat Transfer Part B: Fundamentals,* **28**, 353-369.

Chen, Q., Zhang, Z., and Zuo, W. (2007) "Computational fluid dynamics for indoor environment modeling: Past, present, and future", In: *Proceedings of the 6th International Indoor Air Quality, Ventilation and Energy Conservation in Buildings Conference (IAQVEC 2007)*, Sendai, Japan., 3, pp. 1-9.

Chen, Q.Y. and Xu, W.R. (1998) "A zero-equation turbulence model for indoor airflow simulation", *Energy and Buildings,* **28**, 137-144.

Chorin, A.J. (1967) "A numerical method for solving incompressible viscous flow problems", *Journal of Computational Physics,* **2**, 12-26.

Courant, R., Friedrichs, K., and Lewyt, H. (1928) "Uber die partiellen differenzgleichungen der mathematischen physik", *Mathematische Annalen,* **100**, 32-74.

Courant, R., Isaacson, E., and Rees, M. (1952) "On the solution of nonlinear hyperbolic differential equations by finite differences", *Communication on Pure and Applied Mathematics,* **5**, 243–255.

Crowe, C.T., Troutt, T.R., and Chung, J.N. (1996) "Numerical models for two-phase turbulent flows", *Annual Review of Fluid Mechanics,* **28**, 11-43.

Fedkiw, R., Stam, J., and Jensen, H.W. (2001) "Visual simulation of smoke", *Proceedings of SIGGRAPH 2001*15-22.

Ferziger, J.H. and Peric, M. (2002) *Computational methods for fluid dynamics* (3rd, rev. ed.), Berlin, New York, Springer.

Harris, M.J. (2003) *Real-time cloud simulation and rendering*, Ph.D. Thesis, University of North Carolina at Chapel Hill.

Kim, J., Moin, P., and Moser, R. (1987) "Turbulence statistics in fully-developed channel flow at low Reynolds-number", *Journal of Fluid Mechanics,* **177**, 133-166.

Liu, Y., Liu, X., and Wu, E. (2004) "Real-time 3D fluid simulation on GPU with complex obstacles", In: *Proceedings of 12th Pacific Conference on Computer and Applications (PG'04)*, Seoul, Korea, pp. 247-256.

Megri, A.C. and Haghighat, F. (2007) "Zonal modeling for simulating indoor environment of buildings: Review, recent developments, and applications", *HVAC&R Research,* **13**, 887-905.

Nielsen, P.V. (2004) "Computational fluid dynamics and room air movement", *Indoor Air,* **14**, 134-143.

Restivo, A. (1979) *Turbulent flow in ventilated room*, Ph.D. Thesis, University of London, U. K.

Schaelin, A., Dorer, V., Maas, J.V.D., and Moser, A. (1994) "Improvement of multizone model predictions by detailed flow path values from CFD calculations", *ASHRAE Transactions,* **100**, 709-720.

Scheidegger, C.E., Comba, J.L.D., Da Cunha, R.D., and Corporation, N. (2005) "Practical CFD simulations on programmable graphics hardware using smac", *Computer Graphics Forum,* **24**, 715-728.

Stam, J. (1999) "Stable fluids", In: *Proceedings of 26th International Conference on Computer*

Graphics and Interactive Techniques (SIGGRAPH'99), Los Angeles, pp. 121-128.

USFA. (2007) *Fire statistics*, from http://www.usfa.dhs.gov/statistics/national/index.shtm.

Wang, L. and Chen, Q. (2007) "Analysis on the well-mixing assumptions used in multizone airflow network models", In: *Proceedings of the 10th International Building Performance Simulation Association Conference and Exhibition (Building Simulation 2007)*, Beijing, China, pp. 1485-1490.

Wang, L. and Chen, Q. (2008) "Evaluation of some assumptions used in multizone airflow network models", *Accepted by Building and Environment*.

Zhai, Z., Zhang, Z., Zhang, W., and Chen, Q. (2007) "Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part-1: Observation of prevalent turbulence models", *HVAC&R Research,* **13**.

Zhang, Z., Zhang, W., Zhai, Z., and Chen, Q. (2007) "Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part-2: Comparison with experimental data from literature", *HVAC&R Research,* **13**.