

VALIDATION OF FAST FLUID DYNAMICS FOR ROOM AIRFLOW

Wangda Zuo and Qingyan (Yan) Chen

School of Mechanical Engineering, Purdue University,
West Lafayette 47905, U.S.A.

ABSTRACT

In some emergencies, such as fire or accidental release of chemical/biological agents in buildings, it is very useful to simulate the flow on real time or even faster than real time so that proper measures can be taken to minimize casualties. The traditional computational fluid dynamics (CFD) simulation of fire or transient contaminant transport in buildings is accurate but too time consuming, such as by using unsteady Reynolds averaged Navier-Stokes equations (URANS) and large eddy simulation (LES). On the other hand, multizone flow network modeling is fast, but its accuracy is poor.

Therefore, a new CFD technology, named Fast Fluid Dynamics (FFD), was developed. The FFD is faster than traditional CFD, and more accurate than multizone modeling. This paper shows the validation of the FFD through three cases: (1) flow in a lid-driven cavity; (2) flow in a plane channel; and (3) flow in a ventilated room. The results conclude that the FFD method can simulate the flows faster than real time, although some discrepancies exist between the numerical results and experimental data. The discrepancies are acceptable for the emergency management.

KEYWORDS

Fast Fluid Dynamics (FFD), semi-Lagrangian, real-time simulation

INTRODUCTION

Fire or accidental release of chemical/biological agents in buildings happens occasionally. In such emergent situations, fast prediction of the smoke or contaminant transport is crucial to propose measures to minimize casualties. The prediction should also be accurate and informative.

Unfortunately, none of current modeling technologies can meet the needs. Either their computation speed is too slow or their accuracy is too poor. For example, 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 the unsteady Reynolds averaged Navier-Stokes equations (URANS) is much faster than LES, it still needs a few days of computing time to simulate the airflow in a building. Alternatively, multizone flow network models need little computing time but the accuracy is poor (Wang 2007). However,

the homogenous assumption of airflow in a room does not provide informative results for emergency management. Therefore, it is necessary to develop a method that is faster than the traditional CFD, but more accurate and informative than the multizone modeling.

In weather prediction, it is essential to predict correctly and timely the motion and temperature of atmosphere. By treating the linear terms responsible for gravitational oscillations in an implicit manner, Robert et al. (1972) proposed a semi-Lagrangian scheme, which can increase the time step size by about six times, at little additional cost and without degrading the accuracy of the solution. Since the semi-Lagrangian approach has been successfully used in weather forecast, one might use this method to integrate the Navier-stokes equation and predict the indoor airflow on real time. This forms the fundamentals of FFD presented in this paper.

Furthermore, the semi-Lagrangian approach is mainly applied to climate models in open environment. It is not clear whether FFD would work for indoor environment. Therefore, it is essential to validate the FFD for various flows that have the basic features of indoor airflow.

SCHEME OF FAST FLUID DYNAMICS

FFD solves directly the Navier-Stokes equations:

$$\frac{\partial}{\partial t} U_i = -U_j \frac{\partial}{\partial x_j} U_i + \nu \frac{\partial^2}{\partial x_j^2} U_i + \frac{\partial P}{\partial x_j} + f_i, \quad (1)$$

where U_i and U_j are fluid velocity component in x_i and x_j direction, respectively; ν is kinematic viscosity; P is pressure; and f_i is force sources. Applying the Eulerian approach to species concentration, then the FFD solves

$$\frac{\partial}{\partial t} C = -U_j \frac{\partial}{\partial x_j} C + k \frac{\partial^2}{\partial x_j^2} C + S, \quad (2)$$

where C is species concentration; k is diffusivity; and S is sources. Those partial differential equations are discretized and can then be solved numerically.

For each time step, the numerical algorithm solves the unsteady, diffusion and pressure terms of the equations using Eulerian approach as traditional CFD does. The algorithm solves the advection term by the semi-Lagrangian approach proposed by Robert et al. (1972). An implicit time difference scheme is applied

to ensure that the simulation is stable with an arbitrary time step size.

VALIDATION OF FAST FLUID DYNAMICS

This investigation used three selected cases to validate the FFD method: (1) flow in a lid-driven cavity; (2) flow in a plane channel; and (3) flow in a ventilated room because these flows have the basic features of indoor airflow.

Flow in a lid-driven cavity

Flow in a lid-driven cavity as shown in Figure 1 is like circulated airflow in a room. Because its geometry and boundary conditions are simple, the use of this case can minimize errors. A bunch of experimental and numerical simulation data is available for this case (Bozeman and Dalton 1973; Erturk et al. 2005; Ghia et al. 1982).

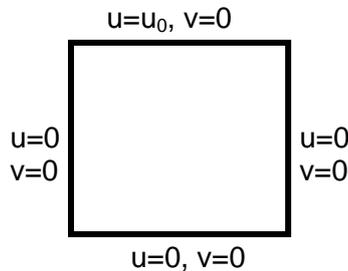
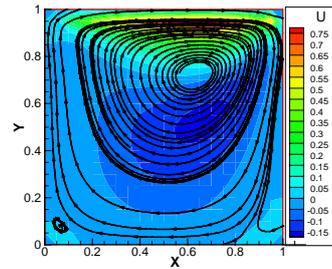


Figure 1 Sketch of lid-driven cavity case

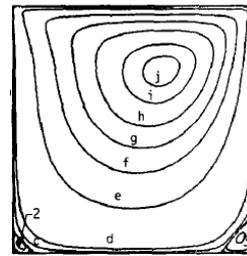
Thus, this investigation used the lid-driven cavity flow case for validation. Figure 2 and 3 compare the FFD results with data from Ghia et al. (1982) for $Re = 100$ and 1000 . Ghia et al. obtained the data by solving the vorticity-stream function of two-dimensional incompressible Navier-Stokes equations. Their solution was 129×129 grid cells but ours 20×20 cells.

Figure 2 shows the streamlines estimated by the FFD and from Ghia et al. (1982) for $Re = 100$. The FFD can predict the main recirculation and the shape of the streamlines is close to the data. For such flow, small recirculations will start to appear at the corners of the cavity, as Reynolds number increases. This is an important feature for validation. The FFD can calculate small recirculations at the lower corners.

Figure 3 compares the streamlines at $Re = 1000$. Although the FFD can predict the main recirculation, its center moves slightly to the right-up corner. The prediction of the small corner recirculations is less satisfactory, because integrating the advection term by semi-Lagrangian approach is more accurate when

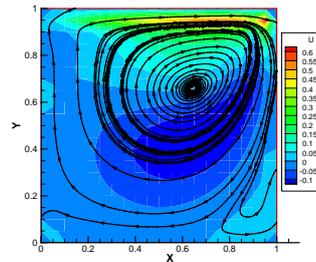


(a) FFD



(b) Ghia et al. (1982)

Figure 2 The predicted streamlines at $Re=100$ for the flow in a lid-driven cavity



(a) FFD



(b) Ghia et al. (1982)

Figure 3 The predicted streamlines at $Re=1000$ for the flow in a lid-driven cavity

the flow is laminar than turbulent. That is why the FFD method works better for the flow at $Re = 100$.

When the Reynolds number is high, one has to use smaller grid cells and shorter time steps in order to obtain accurate solution. The two cases used the same grids and time steps, it is not surprised that the FFD prediction is better for $Re = 100$ than for $Re = 1000$. Although not showing here due to limited space, our further test showed that the FFD could achieve better results if more grid cells and shorter time steps are applied to the case with $Re = 1000$.

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 the second validation case.

Based on the bulk mean velocity, U_b , and the channel half-width, H , the flow Reynolds number studied is 2800. 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 32×8 uniform grids. Figure 4 compares the normalized mean streamwise velocity obtained by the FFD and the DNS data. The FFD under-predicted the velocity at the near wall region. At the channel center, the prediction agrees better with the DNS data.

The coarse grid near the wall is a possible reason led to this discrepancy. The corresponding y^+ is 45, which is far away from the wall. The no-slip wall boundary condition is not valid at this distance. Therefore, a wall function or finer grid may improve the result.

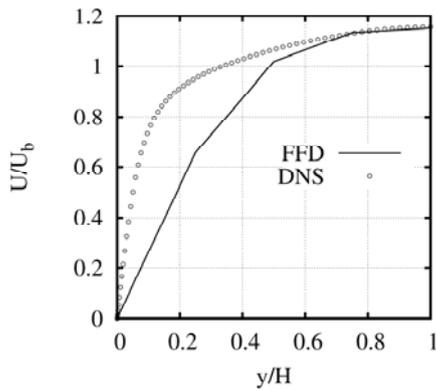


Figure 4 The comparison of mean streamwise velocity of the plane channel flow at $Re = 2800$, predicted by FFD and DNS (Kim et al. 1987)

Flow in a ventilated room

The third case is airflow in an empty, ventilated room that is very close to reality. Restivo (1979) measured the flow as shown in Figure 5, where $H = 3$ 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 300×125 uniform grid cells. Figure 6 shows the experimental data and its comparison with the FFD results in two horizontal and two vertical lines across the room.

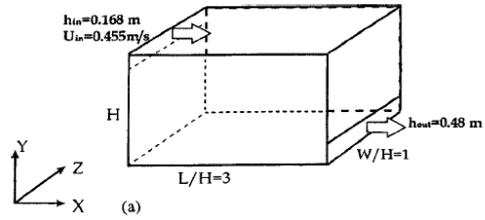


Figure 5 The sketch of empty room with ventilation

The data illustrates that the flow was complex because there were a secondary recirculation in the upper-right corner and another in the lower-left corner. The agreement between the FFD results and the data is not super accurate but acceptable for room air distribution design.

Figure 7 further compares the simulated results by the FFD with those by the standard $k-\epsilon$ model (Chen 1995) and the LES simulation (Su et al. 2001). The FFD can successfully calculate the two secondary recirculations. The $k-\epsilon$ model cannot predict the two recirculations. The LES can only simulate the secondary recirculation at the right-upper corner, but not the one in the lower-left corner. The FFD performance is surprising, although the reason is unknown yet.

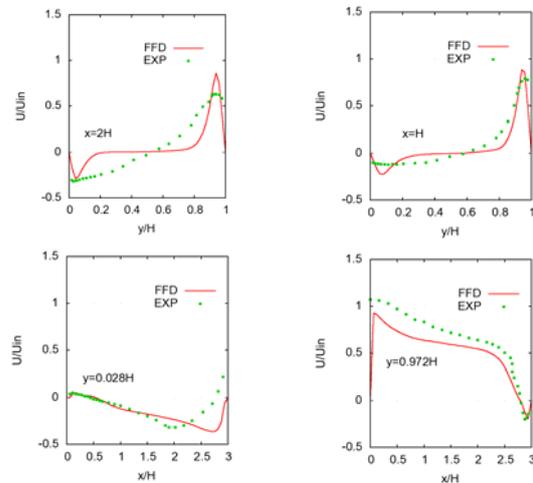
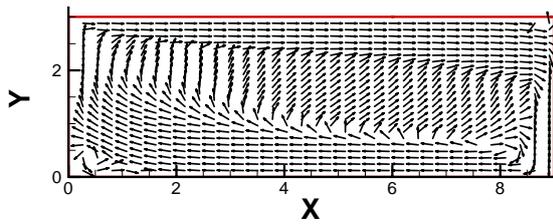


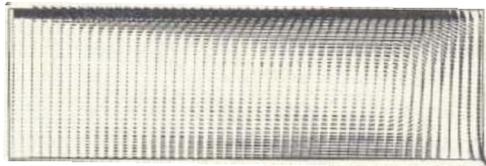
Figure 6 Comparison of the mean horizontal velocities by the FFD with the data (Restivo 1979) at two vertical and two horizontal sections across the room

DISCUSSION

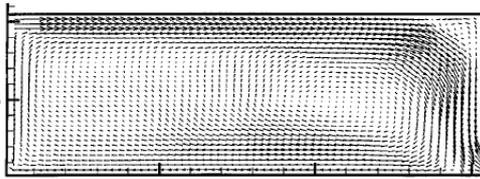
This investigation also evaluated the computing speed of the FFD method. The evaluation defined a “speed enhancement” as $N = t_{physical} / t_{cpu}$, where t_{cpu} is the elapsed CPU time used by the FFD and $t_{physical}$ is physical time of flow motion. Thus, real time simulation is achieved when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



(a) FFD



(b) Standard $k-\epsilon$ model (Chen 1995)



(c) LES (Su et al. 2001)

Figure 7 The velocity field predicted by different numerical methods.

The N strongly depends on number of grids and time step size. 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. For the three cases, the FFD simulations were faster than the real time on a Dell Inspiron laptop with an Intel Core 2 CPU T200 at 2.00 GHz. Table 1 lists the performance of the FFD simulations. Although this CPU is dual core, the FFD simulations used only one processor.

Table 1 Performance of FFD

CASE	GRIDS	Δt (s)	N
Lid-driven cavity	20×20	0.1	44.5
Plane channel	32×8	0.05	30.3
Ventilated room	300×125	0.5	2.4

CONCLUSION

The Fast Fluid Dynamics (FFD) method based on semi-Lagrangian method was validated for three different flows: flow in a lid-driven cavity, flow in a plane channel, and flow in a ventilated room. The accuracy of the FFD method has been evaluated by comparing the predicted results with the experimental and reference CFD data. The FFD method can predict the flow with acceptable accuracy at a speed faster than the real time.

ACKNOWLEDGMENT

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

- Bozeman JD. and Dalton C. 1973. "Numerical Study of Viscous Flow in a Cavity," *Journal of Computational Physics*, 12(3): 348-363.
- Chen Q. 1995. "Comparison of Different K-Epsilon Models for Indoor Air-Flow Computations," *Numerical Heat Transfer Part B-Fundamentals*, 28(3): 353-369.
- Erturk E, Corke TC, and Gokcol C. 2005. "Numerical solutions of 2-D steady incompressible driven cavity flow at high Reynolds numbers," *International Journal for Numerical Methods in Fluids*, 48(7): 747-774.
- Ghia U, Ghia KN, and Shin CT. 1982. "High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method," *Journal of Computational Physics*, 48(3): 387-411.
- 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.
- Restivo A. 1979. "Turbulent flow in ventilated room," Ph.D. Thesis, University of London (U.K.).
- Robert A, Turnbull C, and Henderso J. 1972. "Implicit Time Integration Scheme for Baroclinic Models of Atmosphere," *Monthly Weather Review*, 100(5): 329-335.
- Su M, Chen Q, and Chiang C. 2001. "Comparison of different subgrid-scale models of large eddy simulation for indoor airflow modeling," *Journal of Fluids Engineering-Transactions of the ASME*, 123(3): 628-639.
- Wang L. 2007. "Coupling of Multizone and CFD Programs for Building Airflow and Contaminant Transport Simulations," Ph.D. Thesis, Purdue University.