IMPROVEMENTS ON THE FAST FLUID DYNAMIC MODEL FOR INDOOR AIRFLOW SIMULATION

Wangda Zuo[1,*], Qingyan Chen[1]
[1]National Air Transport Center of Excellence for Research in the Intermodal Transport Environment (RITE), School of Mechanical Engineering, Purdue University, West Lafayette, Indiana, USA
*Current address: Lawrence Berkeley National Laboratory, Berkeley, California, USA

ABSTRACT

Recently, a Fast Fluid Dynamics (FFD) method was proposed for real-time airflow simulation in buildings. As an intermediate model between nodal models and Computational Fluid Dynamics (CFD), the FFD is fast and informative. This paper reports our efforts on improving the accuracy of FFD model. The linear interpolation in semi-Lagrangian solver for advection equation can cause significant numerical diffusion. To minimize numerical diffusion, we proposed a hybrid interpolation. Our simulation results show that the hybrid scheme can significantly improve the accuracy.

INTRODUCTION

Fast and informative flow simulation can be used for the design of building ventilation system, evaluation of building energy performance, building emergency management and indoor environment control. However, neither nodal models nor CFD can satisfy this requirement. The nodal models are fast, but not informative. The CFD is accurate, but too slow. As an intermediate method between the nodal models and the CFD, a FFD (Stam, 1999) method was proposed to fill the gap. Our primary studies (Zuo and Chen, 2009) on the FFD model showed that it could provide detailed flow information, such as flow velocity, air temperature and contaminant concentration, at a speed 50 times faster than the CFD. Although FFD is not as accurate as CFD, it is sufficiently good for building design or emergency management. Thus, the FFD model has a great potential for use in buildings.

The FFD was originally proposed by Stam (1999) for computer visualization and computer games. It applies a time-splitting technique to separate the Navier-Stokes equations into several simple equations. Then those splitted equations are solved one by one. Thus, FFD can solve the Navier-Stoke equations without a trial-correction iterations used in CFD. This is the reason why FFD is much faster than CFD. However, current FFD model applied many low-order schemes, so its accuracy is poorer than CFD. For instance, because of a linear interpolation used in a semi-Lagrangian approach for advection equation, the FFD has a significant numerical diffusion (Fedkiw et al., 2001).

To reduce the numerical diffusion in the FFD model, improvements have been done in two directions. One is applying a high-order interpolation scheme in the semi-Lagrangian solver. For instance, Fedkiw et al (2001) proposed a monotonic cubic interpolation, while Song et al (2005) adopted a constrained interpolation profile (CIP) (Yabe and Aoki, 1991; Yabe et al., 1991). Both methods were reported to be able to reduce the numerical diffusion in the FFD model. However, the CIP method is very complex since it introduces more than ten extra equations. It also needs additional computer memory to store the gradients of interpolated variables. Thus, this investigation did not use the CIP method but the monotonic cubic interpolation. Unfortunately, the monotonic cubic interpolation did not yield a better accuracy for the flows we have tested.

Instead of seeking higher-order interpolations for the semi-Lagrangian solver, another improvement was to solve the advection equation using the convectional method used in CFD, such as the Lax-Wendroff scheme (Lax and Wendroff, 1960) and the QUICK (Quadratic Upwind Interpolation for Convective Kinetics) (Leonard, 1979). For instance, Molemaker et al (2008) tried to eliminate the numerical diffusion by applying the QUICK scheme. However, both Lax-Wendroff and QUICK schemes are not unconditionally stable. In addition, the QUICK scheme will have numerical dispersion if the profile is not smooth.

Through the literature review, we find that none of current proposals can provide a satisfactory solution for the numerical diffusion problem in the FFD. Considering the FFD should be fast and stable, the semi-Lagrangian solver is still a good choice for the FFD. Thus, the challenge is to seek a better interpolation for the semi-Lagrangian solver. To gain a better understanding about the numerical diffusion, we would start our research with a mathematical analysis of numerical diffusion. Then we will propose our solution according to the analysis.

ANALYSIS OF NUMERICICAL DIFFUSION IN THE FFD MODEL

The FFD model splits the Navier-Stokes equations into following equations:

$$\frac{\phi^{(1)}-\phi^{(n)}}{\Delta t}=-U_j\frac{\partial \phi^{(n)}}{\partial x_j}, \tag{1}$$

$$\frac{\phi^{(2)}-\phi^{(1)}}{\Delta t}=\nu\frac{\partial^2 \phi^{(2)}}{\partial x_j^2}+S, \tag{2}$$

$$\frac{\phi^{(n+1)}-\phi^{(2)}}{\Delta t}=P\cdot \tag{3}$$

where ϕ is a single velocity component, ν is kinematic viscosity, S is source, P is pressure, Δt is time step size, superscript $^{(n)}$ and $^{(n+1)}$ present previous and current time step, $^{(1)}$ and $^{(2)}$ are interval time between $^{(n)}$ and $^{(n+1)}$. The FFD first applies a semi-Lagrangian method (Courant et al., 1952) for solving advection equation (1). Then it computes the diffusion equation (2) by using an implicit scheme. Finally, it solves equation (3) together with continuity equation by using a pressure-correction method (Chorin, 1967). More details about the FFD model were given in our previous paper (Zuo and Chen, 2009).

Studies show that the advection solver in the FFD can cause significant numerical diffusion. It is to be analyzed in this section. The split advection equation (1) can be expanded to

$$\frac{\partial \phi}{\partial t}+U\frac{\partial \phi}{\partial x}+V\frac{\partial \phi}{\partial y}+W\frac{\partial \phi}{\partial z}=0, \tag{4}$$

where U, V, and W are velocity components at x, y, and z directions, respectively.

We can re-write equation (4) using material difference:

$$\frac{D\phi}{Dt}=0\cdot \tag{5}$$

Equation (5) means that if we follow a flow particle, we will see that ϕ on this particle is constant with time. In other words,

$$\phi^{n+1}=\phi^{n}, \tag{6}$$

where n and n+1 represent the previous and current time. This is the basic idea of semi-Lagrangian method.

For simplicity, we use a one-dimensional flow as an example to analyze the numerical diffusion. Similar conclusion can also be made by using two or dimensional flows. We also assume that the grid is uniform and the time step size is:

$$\Delta x=x_i-x_{i-1};\Delta t<\Delta x/U, U>0, \tag{7}$$

where x_i and x_{i-1} are the coordinates of grid points i and i-1.

ϕ_i^{n+1} is an unknown variable at grid x_i. If the particle is currently located at x_i, then we can get its location in the previous time step by using a one step tracing:

$$x=x_i-U\Delta t\cdot \tag{8}$$

Since we have known ϕ^n at all grid points in the previous time step, we can obtain $\phi^n(x)$ with interpolation. The FFD applies a linear interpolation:

$$\begin{aligned}\phi^n(x)&=\frac{U\Delta t}{\Delta x}\phi_{i-1}^n+\frac{\Delta x-U\Delta t}{\Delta x}\phi_i^n\\&=\phi_i^n-U\Delta t\frac{\phi_i^n-\phi_{i-1}^n}{\Delta x}.\end{aligned} \tag{9}$$

Since ϕ on this particle does not change, then we have:

$$\phi_i^{n+1}=\phi^n(x)=\phi_i^n-U\Delta t\frac{\phi_i^n-\phi_{i-1}^n}{\Delta x}\cdot \tag{10}$$

ϕ_i^{n+1} is the solution for equation (4) by using semi-Lagrangian solver. To evaluate the accuracy of ϕ_i^{n+1}, we expand ϕ_{i-1}^n by first using a Taylor series:

$$\phi_{i-1}^n=\phi_i^n-\left(\frac{\partial\phi}{\partial x}\right)_i^n\Delta x+\left(\frac{\partial^2\phi}{\partial x^2}\right)_i^n\frac{\Delta x^2}{2}+O(\Delta x^3)\cdot \tag{11}$$

Substitute equation (11) into equation(10), we have

$$\phi_i^{n+1}=\phi_i^n-U\Delta t\left(\left(\frac{\partial\phi}{\partial x}\right)_i^n-\left(\frac{\partial^2\phi}{\partial x^2}\right)_i^n\frac{\Delta x}{2}+O(\Delta x^2)\right)\cdot \tag{12}$$

Re-arranging equation (12), we obtain

$$\frac{\phi_i^{n+1}-\phi_i^n}{\Delta t}=-U\left(\frac{\partial\phi}{\partial x}\right)_i^n+\frac{U\Delta x}{2}\left(\frac{\partial^2\phi}{\partial x^2}\right)_i^n+O(\Delta x^2)\cdot \tag{13}$$

Finally, equation (13) can be formatted as

$$\frac{\partial\phi}{\partial t}+U\frac{\partial\phi}{\partial x}=\frac{U\Delta x}{2}\frac{\partial^2\phi}{\partial x^2}+O(\Delta x^2)\cdot \tag{14}$$

Equation (14) is a modeled equation of equation (4) and it is the actual equation computed by the semi-Lagrangian approach. Compared to equation (1), the modeled equation has an extra second-order term $0.5U\Delta x\partial^2\phi/\partial x^2$. This term is analogous to a diffusion term $\nu\partial^2\phi/\partial x^2$ in the momentum equation. Because it is artificially generated by numerical scheme, this term is called “numerical diffusion” or “artificial diffusion”. Accordingly, $0.5U\Delta x$ is named “numerical viscosity”.

Comparing with the kinematic viscosity, the numerical viscosity is significant for indoor airflow simualtions. For instance, a typical indoor air velocity, *U,* should be lower than 0.2 m/s. A typical mesh size, Δx, for indoor airflow simulation, is about 0.1 m. Then the numerical viscosity $0.5U\Delta x$ is about 1×10^{-2} m^2/s, which is 600 times larger than kinematic viscosity of air at room temperature (~1.568×10^{-5} m^2/s). Thus, it is necessary to reduce the numerical diffusion for improving the accuracy of the FFD model.

A HYBRID INTERPOLATION

Since the numerical diffusion is caused by the low-order linear interpolation, it is possible to minimize it by using a higher-order interpolation scheme. We start our investigation with a third-order interpolation:

$$\phi(x)=\phi_i+(x-x_i)\frac{\phi_{i+1}-\phi_{i-1}}{2\Delta x} + \frac{(x-x_i)^2}{2}\frac{\phi_{i-1}+\phi_{i+1}-2\phi_i}{(\Delta x)^2}. \quad (15)$$

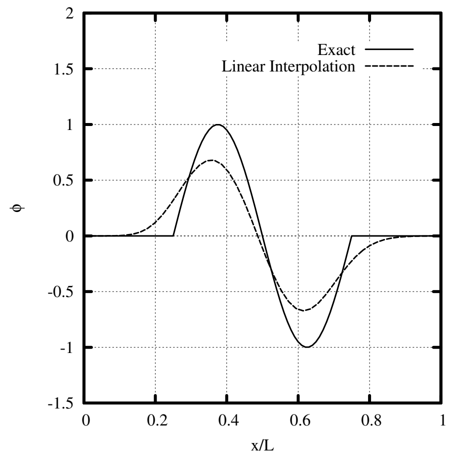
Figure 1 shows the transportation of a sine wave in an inviscid flow. With numerical diffusion, the linear interpolation damped the amplitude of predicted sine wave. Without the numerical diffusion, the third order interpolation can properly catch the sine profile, although it has small error at the feet of since wave.

For a non-smooth square wave (Figure 2), the linear interpolation still computed a smooth profile and the amplitude was under-predicted. On the other hand, the third-order scheme predicted reasonable amplitude, but it had oscillations due to numerical dispersion.

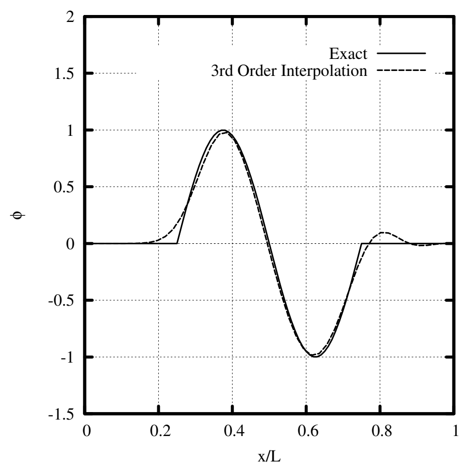
To utilize the merits of both schemes, this study proposed a hybrid interpolation:

$$\phi(x)=\begin{cases}\phi_i+(x-x_i)\dfrac{\phi_{i+1}-\phi_{i-1}}{2\Delta x} + \dfrac{(x-x_i)^2}{2}\dfrac{\phi_{i-1}+\phi_{i+1}-2\phi_i}{(\Delta x)^2} & (\phi_{i+1}-\phi_i)(\phi_{i+1}-\phi_{i-1})>0 \\ \phi_i+(x-x_i)\dfrac{\phi_{i+1}-\phi_i}{\Delta x}, & (\phi_{i+1}-\phi_i)(\phi_{i+1}-\phi_{i-1})\le 0\end{cases} \quad (16)$$

The hybrid interpolation adopts the third order interpolation when the profile is monotonic or smooth. Otherwise, it takes the linear interpolation for the non-smooth part. It applies a simple criterion $(\phi_{i+1}-\phi_i)(\phi_{i+1}-\phi_{i-1})>0$ to judge if the profile is smooth.

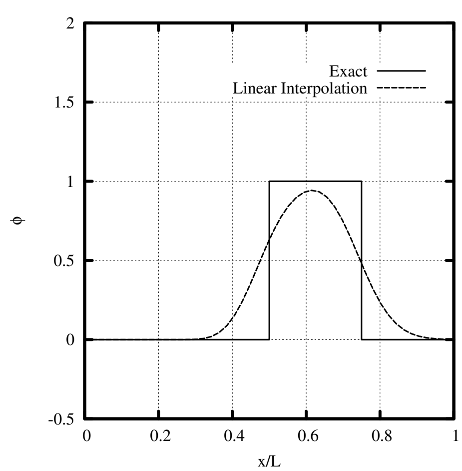


(a) sine wave

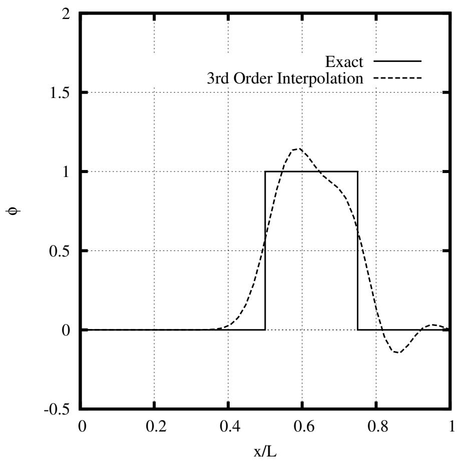


(b) square wave

Figure 1 Comparison of simulated sine waves by linear and third order interpolations



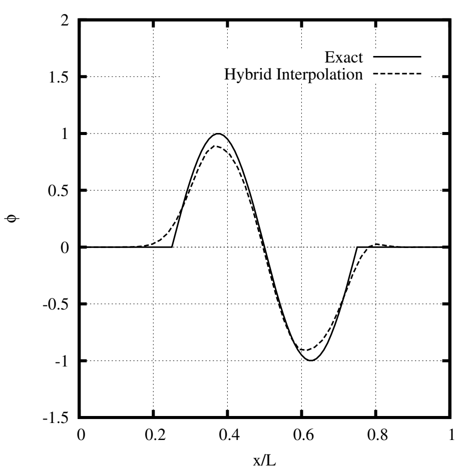
(a) sine wave



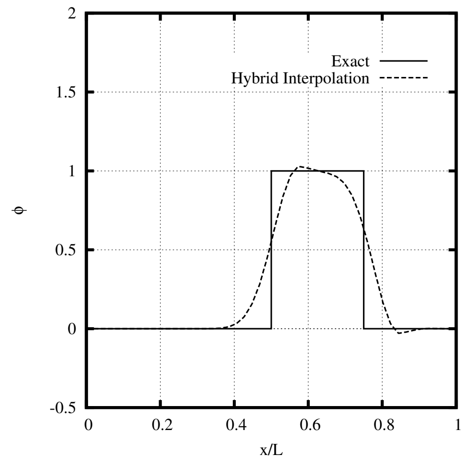
(b) square wave

Figure 2 Comparison of simulated square waves by linear and third order interpolations

As shown in Figure 3, the hybrid interpolation further improved the results by minimizing numerical diffusion and suppressing numerical dispersion. Combining the merits of both linear- and third-order interpolations, the hybrid interpolation seems to be a promising solution for the FFD model. Since the sine and square waves are ideal flows, the results for these flows are not sufficient to assess the performance of hybrid scheme. Thus, we further evaluated the hybrid method by using two cases of complex flows.



(a) Since wave



(b) Square Wave

Figure 3 Comparison of simulated sine and square waves by the hybrid interpolations

SIMULATION RESULTS

Flow in a lid-driven cavity

The flow in a lid-driven cavity is like a circulated flow in a room. Figure 4 is a schematic view of this flow. Based on lid velocity U_0 and square height, L, Reynolds numbers of the studied flows are 100, 1000, and 10000, respectively. With the increase in the Reynolds numbers, the flow transfers from laminar to turbulent. Correspondingly, flow profiles also changes from smooth to non-smooth. Since the FFD model was expected to provide reasonable results on a relatively coarse mesh, it is important to know its performance on different meshes, especially coarse grids. Thus, we applied four different meshes (17 × 17, 33 × 33, 65 × 65, 129 × 129) to this flow to assess the impact of mesh size on the accuracy.

Figure 5 compares the predicted velocity U at vertical mid-section (x = 0.5L) with various Reynolds numbers. All the results were obtained by using a fine mesh of 129 × 129 grids. At Re = 1000 (Figure 5a), the linear interpolation damped the velocity peak values due to the numerical diffusion. With less numerical diffusion, the hybrid interpolation can predict the peak values. The difference increased significantly as the Reynolds number increased to 10000 (Figure 5b).

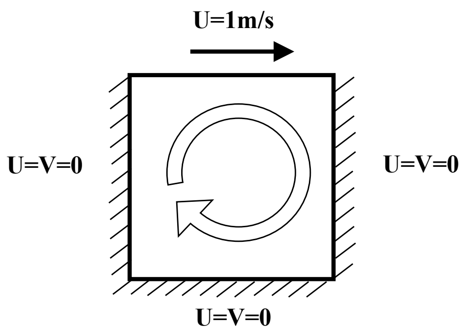
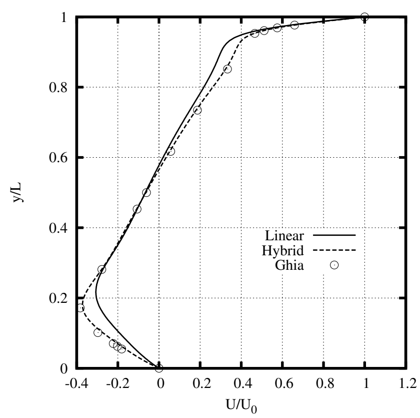
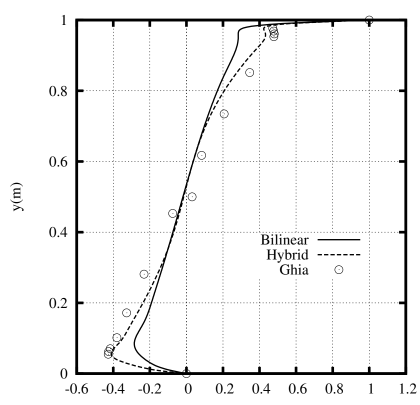


Figure 4 Schematic view of a flow in a lid-driven cavity

The performance of the two schemes with different meshes was quantitatively evaluated by using a root mean square deviation, ϕ_{RMS}. The ϕ_{RMS} represents the total computing error:



(a) U at Re =1000



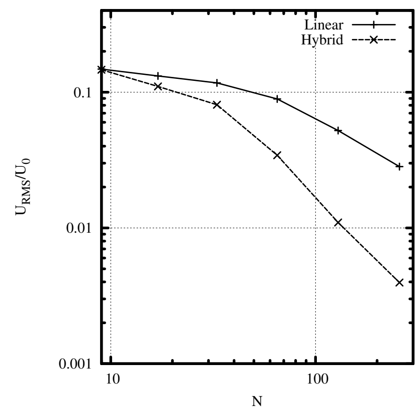
(b) U at Re = 10000

Figure 5 Comparison of velocity profiles at different Reynolds numbers computed by the FFD with linear and hybrid interpolations

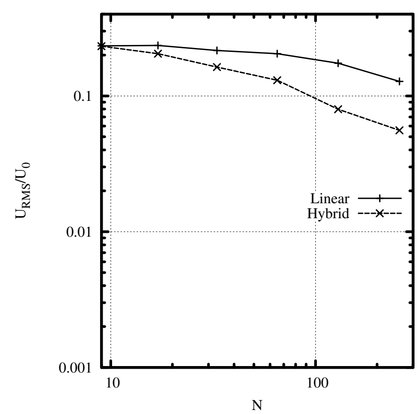
$$U_{RMS} = \sqrt{\frac{\sum_{i=1}^{N}\left(U_i - U_{ref,i}\right)^2}{N}} \tag{17}$$

where N is the total number of sampled data. U_i and $U_{ref,i}$ are the calculated velocity and reference data at location i.

Figure 6 illustrates the relationship between mesh size and root mean square deviation. The U_{RMS} generally decreased when the mesh size increased as one would expect. Figure 6 also shows that the error of the hybrid scheme diminishes faster than that of the linear interpolation. In other words, to obtain the same accuracy, the hybrid interpolation requires fewer grids than the linear one.



(a) U_{RMS} at Re =1000



(b) U_{RMS} at Re = 10000

Figure 6 Comparison of root mean square deviation at different Reynolds numbers computed by the FFD with linear and hybrid interpolations for the lid-driven cavity flow. The grid resolution is N ×N.

Flow over a backward facing step

The airflow which goes into a room through an open window is like a flow passing over a backward facing step. Many experimental data (Armaly et al., 1983; Driver and Seegmiller, 1985; Durst and Schmitt, 1985) and numerical simulations (Barton, 1994; Kim and Moin, 1985; Le et al., 1997; Saldana et al., 2005) are available in the literature, although their geometries and flow conditions may vary case by case.

We selected the experiment by Armaly et al (1983) because their geometry was close to a vertical wall with an open window. Because of high quality, their data was widely used for numerical validation (Das and Devangre, 2009; Lian et al., 2009; Liu, 2009). Figure 7 illustrates the schematic view of their experiment. Height of the step, H, was the half of the channel height. The flow in the small channel was fully developed when it reached the step at $x = 0$. The Reynolds number, Re, was based on the averaged inlet velocity, U_{in}, and step height, H. Armaly et al (1983) found that the flow was laminar when $Re < 1200$ and the flow pattern was two-dimensional when $Re < 400$.

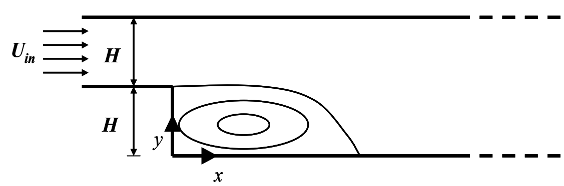


Figure 7 Schematic view of a flow passing the backward facing step

Since the FFD model did not use a turbulence model, it is better to focus on the laminar flow. Thus, we compared both schemes using the flow at $Re = 100$ and 389. To save the computing time, this study omitted the long small channel. Instead, a parabolic profile for laminar flow was given at intersection of small and large channel. Since these flows were two-dimensional, four different meshes (25 × 25, 50 × 50, 100 × 100 and 200 × 200) were applied until a mesh independent solution was achieved.

Table 1 compares the calculated reattachment length from the step with the experimental data (Armaly et al., 1983). All the data was normalized by H and the error was relative to the experimental data. Due to the numerical diffusion, the FFD model computed a shorter reattachment length at the two Reynolds numbers. However, with less numerical diffusion, the FFD with the hybrid interpolation predicted better results than that with the linear interpolation. When the Reynolds number was small ($Re = 100$), the hybrid interpolation predicted the reattachment length precisely with an error of less than 1%. The linear interpolation also got reasonable results with an error of less than 5%. When the Reynolds number increased to $Re = 389$, the error increased for both the interpolation methods, but still acceptable.

Table 1 Comparison of normalized reattachment length computed by the FFD with different interpolation schemes

Re	Experiment	Bilinear	Error	Hybrid	Error
100	2.99	2.87	-4.0%	2.97	-0.7%
389	8.59	7.80	-9.2%	7.95	-7.5%

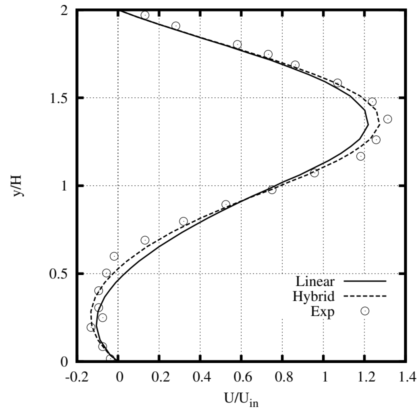
Similar to the lid-driven cavity flow, the hybrid scheme performed better than the linear scheme when the grid was coarse. With finer grids, the predictions of these two methods became closer. For instance, at $Re = 389$, the hybrid interpolation calculated better velocity profiles than the linear one with a coarse mesh of 25 × 25 grids (Figure 8). With a fine of 200 × 200, both methods provided similar results (Figure 9).

CONCLUSIONS

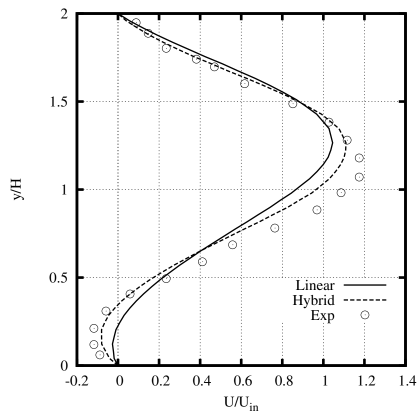
By quantitatively analyzing the interpolation methods in the FFD, this study proposed a hybrid interpolation, which is combination of linear and third order interpolation. The hybrid model can reduce the numerical diffusion in low order model and damp the numerical dispersion in high order model. The validation results showed that the FFD with hybrid interpolation can provide more accurate results than that with linear interpolation. For the same accuracy, the hybrid method requires much less grids and computing time than the linear interpolation does.

ACKONWLEDGEMENT

This study was 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.

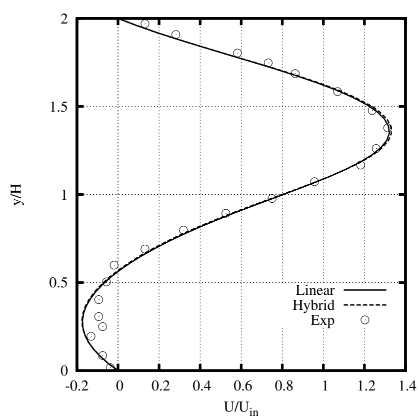


(a) x = 3.57H

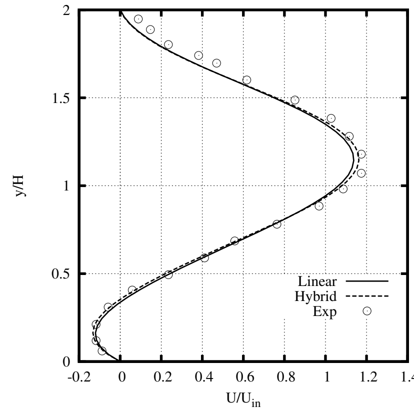


(b) x = 6.12H

Figure 8 Comparison of computed horizontal velocities of backward facing step flow at Re = 389 by using a coarse mesh with 25 × 25 grids.



(a) x = 3.57H



(b) x = 6.12H

Figure 9 Comparison of computed horizontal velocities of backward facing step flow at Re = 389 by using a fine mesh with 200 × 200 grids.

NOMENCLATURE

ϕ_{RMS}	Root mean square deviations
υ	Kinematic viscosity of fluid
Re	Reynolds number
Δt	Time step size
U	Horizontal velocity
V	Vertical velocity
Δx	Mesh size
ϕ	Field variable

REFERENCES

Armaly, B.F., Durst, F., Pereira, J.C.F., and Schonung, B. (1983) "Experimental and theoretical investigation of backward-facing step flow," Journal of Fluid Mechanics, 127, 473-496.

Barton, I.E. (1994) "Laminar-flow past an enclosed and open backward-facing step," Physics of Fluids, 6, 4054-4056.

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

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.

Das, M.K. and Devangre, R.B. (2009) "Conjugate mixed convection heat transfer in plane laminar wall jet flow," Numerical Heat Transfer Part a-Applications, 56, 60-75.

Driver, D.M. and Seegmiller, H.L. (1985) "Features of a reattaching turbulent shear-layer in divergent channel flow," AIAA Journal, 23, 163-171.

Durst, F. and Schmitt, F. (1985) "Experimental studies of high Reynolds number backward-facing step flow," In: Proceedings of the 5th Symposium on Turbulent Shear Flows, pp. 519-524.

Fedkiw, R., Stam, J., and Jensen, H.W. (2001) "Visual simulation of smoke," In: Proceedings of SIGGRAPH 2001, Los Angeles, CA, USA, pp. 15-22.

Kim, J. and Moin, P. (1985) "Application of a fractional-step method to incompressible Navier-Stokes equations," Journal of Computational Physics, 59, 308-323.

Lax, P. and Wendroff, B. (1960) "Systems of conservation laws," Communications on Pure and Applied Mathematics, 13, 217-237.

Le, H., Moin, P., and Kim, J. (1997) "Direct numerical simulation of turbulent flow over a backward-facing step," Journal of Fluid Mechanics, 330, 349-374.

Leonard, B.P. (1979) "Stable and accurate convective modelling procedure based on quadratic upstream interpolation," Computer Methods in Applied Mechanics and Engineering, 19, 59-98.

Lian, C.Z., Xia, G.P., and Merkle, C.L. (2009) "Solution-limited time stepping to enhance reliability in CFD applications," Journal of Computational Physics, 228, 4836-4857.

Liu, J. (2009) "Open and traction boundary conditions for the incompressible Navier-Stokes equations," Journal of Computational Physics, 228, 7250-7267.

Molemaker, J., Cohen, J.M., Patel, S., and Yong Noh, J. (2008) "Low viscosity flow simulations for animation," Symposium on Computer Animation, Retrieved Dublin, Ireland.

Saldana, J.G.B., Anand, N.K., and Sarin, V. (2005) "Numerical simulation of mixed convective flow over a three-dimensional horizontal backward facing step," J. Heat Transfer, 127, 1027-1036.

Song, O.-Y., Shin, H., and Ko, H.-S. (2005) "Stable but nondissipative water," ACM Transactions on Graphics, 24, 81-97.

Stam, J. (1999) "Stable fluids," In: Proceedings of 26th International Conference on Computer Graphics and Interactive Techniques (SIGGRAPH’99), Los Angeles, pp. 121-128.

Yabe, T. and Aoki, T. (1991) "A universal solver for hyperbolic-equations by cubic-polynomial interpolation .1. One-dimensional solver," Computer Physics Communications, 66, 219-232.

Yabe, T., Ishikawa, T., Wang, P.Y., Aoki, T., Kadota, Y., and Ikeda, F. (1991) "A universal solver for hyperbolic-equations by cubic-polynomial interpolation .2. 2-dimensional and 3-dimensional solvers," Computer Physics Communications, 66, 233-242.

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