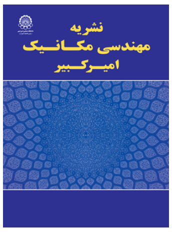


Amirkabir Journal of Mechanical Engineering

Amirkabir J. Mech. Eng., 53(special issue 3) (2021) 443-448
DOI:10.22060/mej.2020.17222.6538

Accurate and faster than real-time simulation of indoor airflow by using fast fluid dynamics

M. Seyedi, H. Pasdarshahri*

Tarbiat Modares University, Faculty of Mechanical Engineering

ABSTRACT: The multizone model is one of the most popular models for simulating indoor energy and airflow; however, its basic problem is that it cannot provide detailed and accurate airflow information. Computational fluid dynamics can be used to obtain detailed airflow information but is restricted due to its high computational cost. Therefore, it is necessary to develop a model which can provide detailed airflow information with reasonable accuracy and computational time. In this study, fast fluid dynamics, which has an unconditionally stable algorithm, is proposed. To investigate this model, four case studies of flow in a lid-driven cavity, flow in a channel, natural and forced flow convention are analyzed, and the results are compared and validated with computational fluid dynamics, experimental data, and analytical solution. The main focus of this study is increasing the computational speed of fast fluid dynamics. For this purpose, the sequence of equations has been modified and suitable numerical methods have been applied to solve each equation. Using the proposed fast fluid dynamics solver, the simulation time of the case studies has decreased between 52 to 94 percent compared to computational fluid dynamics and faster than real-time simulation has been achieved on a conventional computer system.

Review History:

Received: Oct. 15, 2019
Revised:Dec. 30, 2019
Accepted: Mar. 10, 2028
Available Online: Mar. 22, 2028

Keywords:

Fast Fluid Dynamics
Computational Fluid Dynamics
Fluid Flow Simulation
Ventilation, Building Energy Simulation.

1- Introduction

To optimize the Heating, Ventilation, and Air Conditioning (HVAC) systems and reduce energy consumption, a fast airflow simulation model is necessary. Multizone models are widely used in indoor airflow simulations [1]. However, this model fails to correctly predict the airflow where the flow regime is stratified or has high momentum as it assumes homogenous flow characteristics in the room [2]. Computational Fluid Dynamic (CFD) method can be used alternatively to achieve accurate airflow simulation; however, this method is restricted due to its high computational cost. To overcome these issues, Fast Fluid Dynamics (FFD) method can be used as an intermediate approach between multizone and CFD techniques. FFD was firstly introduced by Stam for creating realistic visual effects of fluid flow in computer games [3]. Zuo et al. showed that FFD model has proper accuracy for indoor airflow simulation [4].

The main focus of the present study is increasing the computational speed of FFD by modifying the sequence of equations and using suitable numerical methods for solving each equation. To evaluate the modified FFD model, four case studies of lid-driven cavity flow, flow in a channel, natural and forced flow convection are simulated and the results are validated with experimental data, CFD results, and analytical solution.

2- Methodology

The FFD model is based on the time splitting technique through which the momentum equation (Eq. (1)) is split into four sub-equations (Eqs. (2) to (5)) which are solved successively.

$$\frac{\partial U_i}{\partial t}+U_j\frac{\partial U_i}{\partial x_j}=-\frac{1}{\rho}\frac{\partial P}{\partial x_i}+\nu\frac{\partial^2 U_i}{\partial x_j^2}+f_i \tag{1}$$

$$\frac{\partial U_i^{(1)}}{\partial t}=-U_j\frac{\partial U_i^{(1)}}{\partial x_j} \tag{2}$$

$$\frac{U_i^{(2)}-U_i^{(1)}}{\Delta t}=\nu\frac{\partial^2 U_i^{(2)}}{\partial x_j^2} \tag{3}$$

$$U_i^{(3)}=U_i^{(2)}+\Delta t f_i \tag{4}$$

$$\frac{U_i^{(4)}-U_i^{(3)}}{\Delta t}=-\frac{1}{\rho}\frac{\partial P}{\partial x_i} \tag{5}$$

*Corresponding author's email: pasdar@modares.ac.ir

Other sequences of equations have also been implemented in previous studies; however, the proposed sequence of equations (Eqs. (2) to (5)) was found to be the most efficient among the other possible sequence of equations.

In Eqs. (1) to (5), is velocity in direction, is position component in direction, is kinematic viscosity, is density, is pressure and is volumetric force component in direction. Furthermore, , and are the temporary velocities which are transferred as the initial conditions to the next sub-equation and is the velocity domain in the next time step. Therefore, the complex momentum equation is split into four much simpler equations which can be solved separately with proper numerical methods. In FFD, the convection equation (Eq. (2)) is solved by a semi-Lagrangian approach [5]. which makes the FFD model as an unconditionally stable method. The diffusion equation (Eq. (3)) is solved implicitly by the Gauss-Seidel method. Eq. (4) is solved through simple algebraic operations. To solve Eq. (5) the pressure domain is needed, which is unknown up to this step. In FFD, the coupling between velocity and pressure is solved by pressure projection scheme [6] though which Eq. (5) is coupled with continuity equation (Eq. (6)), which leads to a stiff equation called pressure Poisson equation (Eq. (7)).

$$\frac{\partial U_i}{\partial x_i} = 0 \qquad (6)$$

$$\frac{\partial^2 P}{\partial x_i^2} = \frac{\rho}{\Delta t}\frac{\partial U_i^{(3)}}{\partial x_i} \qquad (7)$$

Using a proper numerical scheme to solve Eq. (7) highly improves the FFD computational speed. This study applied the Alternative Direction Implicit (ADI) method to solve Eq. (7). Therefore by substituting the calculated pressure from pressure Poisson equation into Eq. (5) the velocity domain is calculated and the solver moves on to the next time step.

3- Results and Discussion

2.1.Flow In A Lid-Driven Cavity

The first case study simulated with the proposed FFD model is a lid-driven cavity flow with a Reynolds number of 100 (Fig. 1).

The results of horizontal velocity on axis and vertical velocity on axis are presented in Fig. 2.

The CFD results have been obtained using ANSYS FLUENT software. A 64×64 grid and time step of 0.02 s are implemented in both FFD and CFD simulations. Also, Ghia et al., [7] experimental data have been used for the validation of numerical results. The horizontal and vertical velocity results obtained from the FFD model match well with CFD results and experimental data.

2.2. Natural Flow Convection

The natural flow convection problem with the Rayleigh number of is another case study simulated with the proposed FFD model (Fig. 3).

The temperature results on , and axes are presented in Fig. 4.

A 20×40 grid and time step of 0.05 s are implemented in both FFD and CFD simulations. Also, Betts et al., [8] experimental data have been used for the validation of numerical results. The temperature results agree well with the experimental measurements.

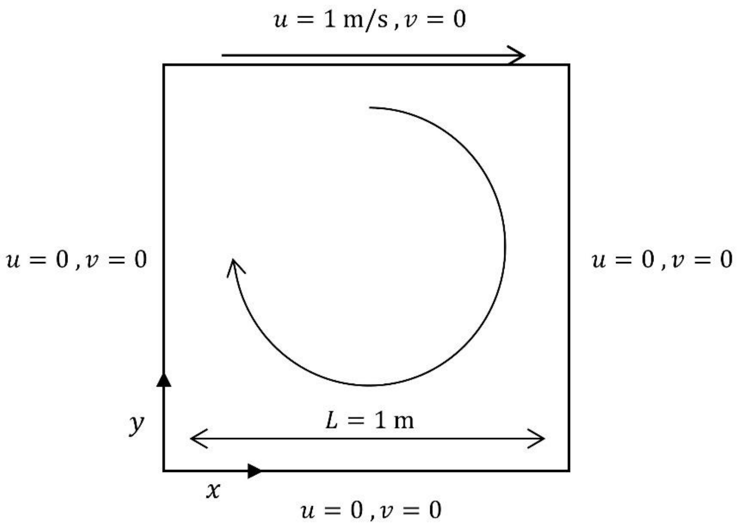
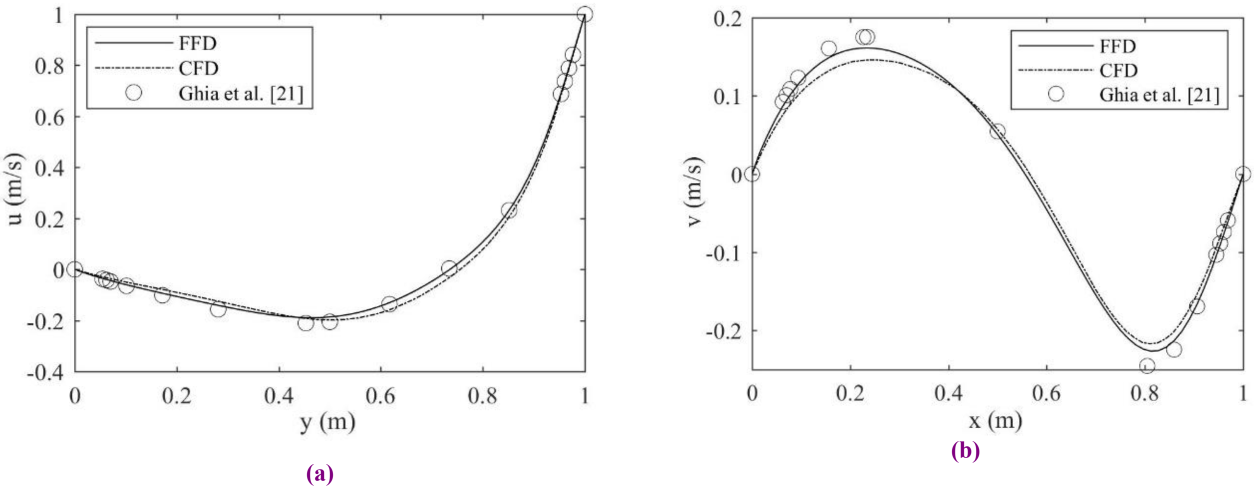


Fig. 1. Geometry and boundary conditions of lid-driven cavity flow problem



(a)

(b)

Fig. 2. Simulation results of lid-driven cavity flow problem with Re=100 (a) horizontal velocity (b) vertical velocity

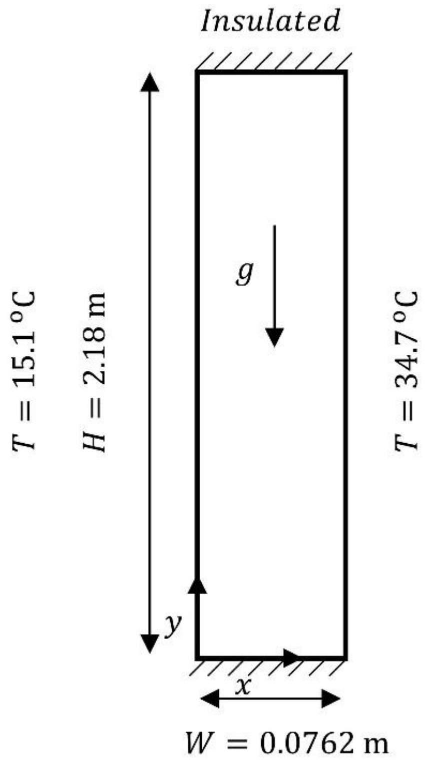
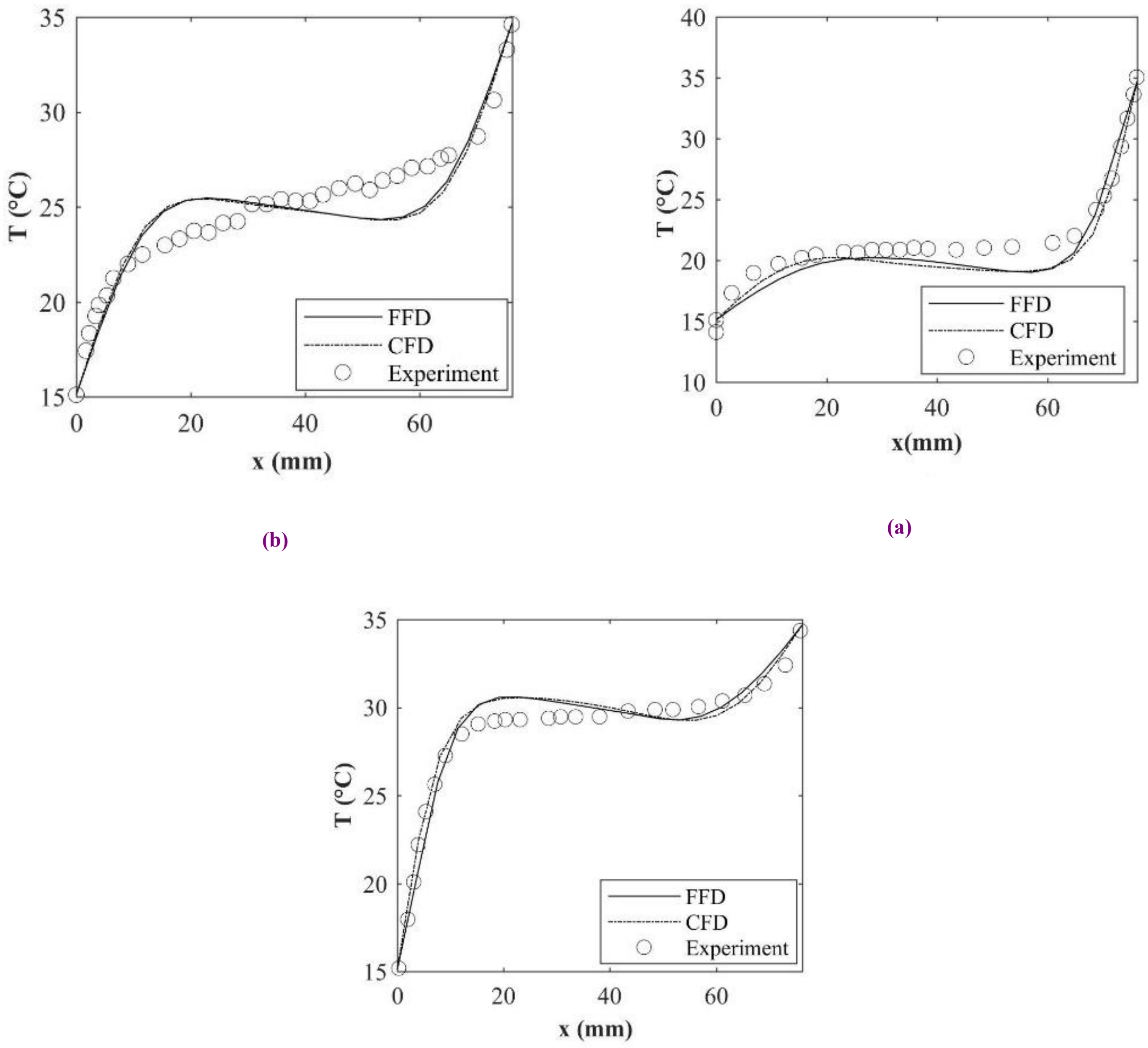


Fig. 3. Geometry and boundary conditions of natural flow convection problem



(b)

(a)

(c)

Fig. 4. Temperature results of natural flow convection problem (a) y=0.218 m (b) y=1.090 m (c) y=1.926 m

In addition to the above problems, the case studies of lid-driven cavity flow with , channel flow with and forced flow convection with have also been simulated with the proposed FFD model and validated with CFD results, analytical solution, and experimental data. In all cases, the FFD computational time was not only faster than real-time but also 52 to 94 percent faster than CFD simulation.

4- Conclusions

In this paper, a modified FFD model with a revised sequence of equations and implementing proper numerical methods for solving each equation was proposed and validated with CFD results, analytical solution, and experimental data. In all cases, the FFD simulation on a regular computer was faster than real-time and also 52 to 92 percent faster than CFD simulation. Three prominent features of FFD, i.e., fast computations, proper accuracy, and an unconditionally stable algorithm, make this model a potential method to be widely used in a variety of building energy simulation problems, which demands more research and development.

REFERENCES

[1] J. Axley, Multizone Airflow Modeling in Buildings: History and Theory, HVAC&R Research, 13(6) (2007)

907-928.

[2] W. Tian, X. Han, W. Zuo, M.D. Sohn, Building energy simulation coupled with CFD for indoor environment: A critical review and recent applications, Energy and Buildings, 165 (2018) 184-199.

[3] J. Stam, Stable fluids, in: Proceedings of the 26th annual conference on Computer graphics and interactive techniques, ACM Press/Addison-Wesley Publishing Co., 1999, pp. 121-128.

[4] W. Zuo, Q. Chen, Real-time or faster-than-real-time simulation of airflow in buildings, Indoor air, 19(1) (2009) 33.

[5] R. Courant, E. Isaacson, M. Rees, On the solution of nonlinear hyperbolic differential equations by finite differences, Communications on Pure and Applied Mathematics, 5(3) (1952) 243-255.

[6] A.J. Chorin, A numerical method for solving incompressible viscous flow problems, Journal of computational physics, 2(1) (1967) 12-26.

[7] U. Ghia, K.N. Ghia, C. Shin, High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method, Journal of computational physics, 48(3) (1982) 387-411.

[8] P. Betts, I. Bokhari, Experiments on turbulent natural convection in an enclosed tall cavity, International Journal of Heat and Fluid Flow, 21(6) (2000) 675-683.

HOW TO CITE THIS ARTICLE

M. Seyedi, H. Pasdarshahri, Accurate and faster than real-time simulation of indoor airflow by using fast fluid dynamics. Amirkabir J. Mech. Eng., 53(special issue 3) (2021). 443-448.

DOI: 10.22060/mej.2020.17222.6538