A Three Layered Framework for Annual Indoor Airflow CFD Simulation (Part II)

Yue Wang[1,*], Ali Malkawi[1], Yun Kyu Yi[1], Ning Feng[1]

[1]T.C. Chan Center for Building Simulation and Energy Studies, University of Pennsylvania, PA, 19104, USA

ABSTRACT
In (Yue Wang, 2012) a three layered framework to optimize CFD to cover annual hourly simulation is proposed. The framework contains the mathematical layer, the model layer, and the application layer. Many of the literature related to GPGPU optimization to speed up the mathematical layer was reviewed based on the framework. In this literature review, the fast fluid simulation algorithm will be employed in the second layer. It can be seen from the review that some techniques can dramatically improve the performance of the second layers; however, a good method has not been found yet for the third layer, which requires more work in the future. It is expected that the annual simulation with this method will use a similar simulation time compared to the traditional two or three extreme cases.

KEYWORDS
CFD, FFD, GPGPU, Machine Learning, Annual Hourly Simulation, Building Simulation

INTRODUCTION
Annual hourly simulation is important in many applications in the building simulation field. Fast indoor airflow simulations are necessary for designing building emergency management systems, the preliminary design of sustainable buildings, and modeling real-time indoor environment control. However, CFD computation is usually time consuming and unable to simulate annual indoor air movement. Thus only one or two extreme cases (like the hottest hour in the summer and coldest hour in the winter) will be simulated by CFD in most of the design analysis. In (Yue Wang 2012) a three layered framework to optimize CFD to cover annual hourly simulation is proposed. The framework contains the mathematical layer, the model layer, and the application layer. Many of the literature related to GPGPU optimization to speed up the mathematical layer was reviewed based on the framework.

This research continues to investigate the model layer and the application layer, and provide a possible implementation for the three layered framework.

[*] Corresponding author email: yuleopen@gmail.com

THE MODEL LAYER

The foundation of CFD is based on the Navier-Stokes equations. The Reynolds-averaged Navier-Stokes (RANS) equations are time-averaged equations of motion for fluid flow. They are primarily used while dealing with turbulent flows. The K-epsilon model is one of the most common RANS turbulence models. It is a two-equation model, which means, it includes two extra transport equations to represent the turbulent properties of the flow. This allows a two-equation model to account for history effects like convection and diffusion of turbulent energy. The model is widely used in building science research, especially indoor air quality and thermo distribution simulation (Cheng W.C. et al, Sun, Huawei et al 2007, Tominaga Yoshihide et al, P. Neofytou et al 2006, R. Panneer Selvam). The K-epsilon model should be a reference implementation for us since it is the most widely used solver in building science. When we developed a new solver, we should compare the results to the K-epsilon model and see whether the new solver is accurate enough (Adrian J.Lew et al). There are three major alternatives to the traditional way, all used heavily in computer animation. They are the Lattice Boltzmann Method, Smooth Particle Hydrodynamics, and Fast Fluid Dynamics.

Instead of solving the Navier-Stokes equations, Lattice Boltzmann methods (LBM) simulate the flow of a Newtonian fluid with collision models such as Bhatnagar-Gross-Krook (BGK) (Sukop, Michael C. et al 2007, Succi, Sauro 2001, Alexander J. Wagner). Due to its particulate nature and local dynamics, LBM has several advantages over other conventional CFD methods, especially in dealing with complex boundaries, incorporating of microscopic interactions, and parallelization of the algorithm (Shiyi Chen and Gary D. Doolen 1998). An important advantage of lattice Boltzmann models allow efficient parallelization of the simulations. Benchmark shows that LBM on parallel machines with fast internet connection have close to linear speed up when more processing units are added (Anon E). However, in general, room airflows are not fully turbulent. Baker states that most room air flows are locally turbulent (Roy, S. et al 1994). Although measurements indicate that the flow in the main body of ventilated rooms may be transitional, air flow at diffuser outlets tends to be turbulent (jones P J and Whittle G E 1992). In this case, it's really hard to use the Lattice Boltzmann Method to catch the turbulent property of indoor airflow, which is not good at simulating low Reynolds number problems.

Another alternative is Smoothed Particle Hydrodynamics (SPH). SPH was developed by L.B.Lucy(1977) and R.A.Gingold(1977) for the simulation of astrophysical problems, the method is general enough to be used in any kind of fluid simulation. For introductions to SPH we refer the reader to J.J.Monaghan(1992) or S.A.Munzel(1996). The use of particles instead of a stationary grid simplifies these two equations substantially. Because the number of particles is constant and each particle has a constant mass, mass conservation is guaranteed and the equation of mass conservation can be omitted completely.

However, even though SPH is a heated topic, the major three problems make it hard to be applied to building airflow simulations. First, in order to simulate realistic chaotic fluids, large amounts of particles should be introducing into the system which should be extremely high. Second, there are extreme difficulties when introduce the traditional turbulent models into the kernel function, since turbulent models usually take advantage of time averages behavior, while simulating the particle movement over a certain amount of time and taking the average behavior will use much more computing power. Third, and the most important, particles in the geometry is always moving similar to the fluid. It is not possible, for example, to calculate a steady-state case, like the traditional one does. This is because the traditional CFD uses Euler view and SPH uses Lagrangian view.

The research thus will use the third approach, Fast Fluid Dynamics, which simplifies the Navier-Stokes equation numerical solving method to make CFD calculations faster, as the foundation of the model layer. In Stam's (1999) paper, for the first time, an unconditionally stable model which still produces complex fluid-like flows was proposed. The unconditionally stable behavior increases the FFD model's performance because it is possible to use arbitrary long time step to perform the simulation. The FFD scheme was originally proposed for computer visualization and computer games (Harris MJ 2003, Song O-Y et al 2005).

FFD had a successful application in computer graphics, which intends to create an appealing effect. The accuracy of FFD for the engineering application is open to question. Some of the recent papers apply FFD to building simulation. Zuo and Chen did initial work to validate FFD for room airflow to Building Simulation Conference 2007 (Zuo, W and Chen, Q). They pursued their research further by publishing a comprehensive conclusion on Indoor Air in 2009 (Zuo, W and Chen, Q). Even though FFD can correctly predict the laminar flow, it has some problems in computing turbulent flows due to the lack of turbulence treatments. Zuo and Chen continue to make a lot of explorations and found many ways to improve their model and released a series of proceeding papers (Zuo, W and Chen, Q 2011, Zuo, W and Chen, Q 2010).

Zuo, W et al(2010) developed the FFD by improving its speed and accuracy. Enhancement of the computing speed can be realized by modifying the time-splitting method. Using the new FFD model for several indoor air flows, the results show a significant reduction in computing time and great improvements on accuracy. The research (Zuo, W et al) further improved the FFD method by reducing the numerical viscosity that is caused by a linear interpolation in its semi-Lagrangian solver. The results show that the hybrid interpolation can significantly improve the accuracy of the FFD with a small amount of extra computing time. In Stam's FFD model, the simulation will almost never reach a steady-state since the simulation numerical scheme does not ensure steady-state convergence. This research requires steady-state solution. In order to solve this problem, the time-averaged method is reintroduced in this research. Flows are simulated for a period of time, and averaged temperature and velocity are treated as steady state results.

THE APPLICATION LAYER

Since annual hourly simulations have more than 8000 hourly cases, to calculate each case independently may take longer simulation time. It is possible to reuse previous calculated results to generate new results, thus considerably shortening the simulation execution time. Among many methods, machine learning algorithms, which use existing results (training and testing set) to train the model which is used to predict future results, seem to be the best option. Notably, artificial neural network (ANN) has been universally utilized. ANN has the solid ability of self-learning and self-organization. It can employ the prior acquired knowledge to respond to the new information rapidly and automatically. In the area of fluid flow, the applications of the ANN include the simulation of fluid flow and transport (Jahangir and Jagath 1998), the calculation of coefficients of heat transfer in fluid-particle systems (S.S.Shyam 2001), the prediction of flow field using a hybrid scheme of ANN (Bening, T.M.Becker and A.Delgado 2001), the dynamics simulation of a steam generator of nuclear reactor (R.Masini et al 1999), and the computation of the friction factor in pipeline flow (Walid and Shyam 1998).

An attractive attribute of machine learning models is their fast speed of prediction computation. Once an ANN model has been trained, tested and validated, it can provide almost real-time parametric and sensitivity analysis. The typical time taken for an ANN model to execute one run is generally several orders of magnitude smaller than that required for running a CFD model (A.S.Kelkar et al 1996). So many researchers are trying to find a numerical method based on ANN to solve NS equations (SMAOUI Nejib and AL-ENEZI Suad 2004, Mai-duy Nam and Tran-cong Thanh 2001, William, P.N., T. Kozek and T. Roska, Nejib Smaoui and Ali A. Garrouch 1997). Most of these works showed that the trained ANN model predicts the behavior of the reactor accurately. The results demonstrate the power and robustness of ANNs for obtaining fast responses to changing input conditions. However, though there is plenty of research focused on using artificial neural network to solve the flow transport problem, most papers, as listed above, do not use the Navier-Stokes equation as its training foundation equation. Most papers have a limited focus field like flow and contaminant transport (GFCT) simulations, simulating Biot number/heat transfer coefficient. Those who use Navier-Stoke equations, usually focus on very low (less than 10) *Re* numbers, thus not applicable to the building simulation field.

Some recent studies also use meshless methods as an alternative to grid-based flow computation (T. Belytschko et al 1996). Recent work has indicated that accurate results may be obtained with meshless methods as compared with grid-based methods (Aaron Katz and Antony Jameson 2008, El Zahab, Z. et al 2009). In 2005, Zhi Shang(2005) combined ANN and meshless simulation together to simulate vapor-water two-phase flows in a tube with uniform and non-uniform heating. It does not generate mesh in the calculation domain except for some random points, nor does it solve the algebraic equations. Again, it's hard for machine learning algorithms to catch the chaos properties of turbulent. Moreover, some machine learning algorithms would not solve the performance problems at all. Usually training all the parameters requires 1,000,000 iterations, which might take a considerable amount of time, and takes much larger computation expanses than an annual result simulation. Thus indoor air is chaos and machine learning algorithm cannot catch the pattern very well.

Another way to reuse the result of existing cases is to use CFD to converge new case simulation using existing cases' results as the starting point, and in some conditions the convergence will take much fewer iterations. It turns out that although the machine learning methods might not capture the chaotic property of indoor airflow well, it can capture some of the other properties well, such as the difference of input of two cases and the iteration steps needed from one case as a string point to converge to the other case. There is a strong relationship between the input variance and the output variance, so is strong relationships between the output variance and the iterations required for convergence from one case to another. This gives a clue to design an efficient algorithm to reuse as many existing simulated cases as possible. The full algorithm will be published as a companion paper.

CONCLUSION

In this paper, we enumerate many different research methods conducted to make CFD much faster. There are three different approaches to speed up the simulation. The solution will follow the layered view, and tackle one layer above another, and gain tremendous speed-up by combining improvements on three layers together.

Table 3. *Three Layered framework. (The estimated total speed-up is expected to be 1000-8000x)*

Layer	*Comments*	*Speed Ups*
Application layer	reusing simulated cases	10-20x
Model Layer	faster turbulent models	10-20x
Math Layer	GPGPU optimization	10-20x

All these methods are integrated and form a complete framework to solve the problem. However, any of these methods should be standalone as well. This gives great portability of this framework, and applying any of these methods becomes more flexible.

1. This research employed techniques from the GPGPU computing world, modifying the low level mathematical procedures to speed up the simulation. This approach can and should be generic enough to be applied to different computational models (including FFD, k-epsilon, Large Eddy Simulation, etc) and achieve tremendous speed up over the traditional CPU method. 2. This research also incorporated Qingyan Chen's FFD model with slight modification to perform steady case analysis. The time

averaged approach will be reincorporated into Qingyan Chen's realtime dynamic simulation FFD model and have the averaged behavior of indoor air flow. 3. By utilizing the fact that many cases are similar in annual simulation, this research proposed the idea that can be utilized to implement an efficient algorithm by reusing existing results as much as possible.

With the three layered optimization strategies workflow, this research is able to simulate annual hourly conditions as fast as doing two extreme cases using traditional approaches by multiplying the time factors of the three. Thus, annual approximate simulation is made possible within a very short period of time, usually within an hour or several hours. All the methods are generally purposed and can be act as an optional plugin. It is possible to just apply one or two methods if the simulation requires high accuracy by leaving only GPGPU computing and optimization strategies in this chapter. It's also possible to eliminate the GPGPU computing power if the case work size is very small, where the GPGPU method may be inefficient in such cases. Also there are other variations to this method. For example, if the case problem size is relatively small, the GPGPU method may fail to achieve in this case. But it is possible to simulate the annual cases altogether since each hour's case's algorithm is exactly as other hour's case's, and thus can be highly parallelized to achieve great speed.

In a nutshell, there is large amount of paper that contributes to one of the three layers; however, currently the biggest problem is to solve and validate these models for building simulation, and integrate all these layers together to maximize the simulation performance.

REFERENCES

Yue Wang, Ali Malkawi, Yun Yi, Ning Feng, A Three Layered Framework for Annual Indoor Airflow CFD Simulation (Part I), The 1st Asia conference of International Building Performance Simulation Association, 2012

CHENG W. C., LIU Chun-Ho, LEUNG Dennis Y. C., Computational formulation for the evaluation of street canyon ventilation and pollutant removal performance

Sun, Huawei, Zhao, Lingying, Zhang, Yuanhui, Evaluating RNG k-[epsilon] models using PIV data for airflow in animal buildings at different ventilation rates. ASHRAE Transactions, January 1, 2007

TOMINAGA YOSHIHIDE, MOCHIDA AKASHI, et, al., Journal of Architecture, Planning and Environmental Engineering, Comparison of performance of various revised k-.EPSILON. models applied to CFD analysis of flowfield around a high-rise building.

P. Neofytou, A.G. Venetsanos, et, al., CFD simulations of the wind environment around an airport terminal building, Environmental Modelling \& Software, Volume 21, Issue 4, April 2006, Pages 520-524

R. Panneer Selvam, Computation of flow around Texas Tech building using k-epsilon and Kato-Launder k-epsilon turbulence model, Engineering Structures Volume 18, Issue 11, November 1996, Pages 856-860

Adrian J. Lew, et, al. A note on the numerical treatment of the k-epsilon turbulence model. to appear in International Journal of Computational Fluid Dynamics.

Sukop, Michael C. and Daniel T. Thorne, Jr. (2007). Lattice Boltzmann Modeling: An Introduction for Geoscientists and Engineers. Springer. ISBN 9783540279815.

Succi, Sauro. The Lattice Boltzmann Equation for Fluid Dynamics and Beyond. Oxford University Press. ISBN 0198503989. 2001

Alexander J. Wagner, A Practical Introduction to the Lattice Boltzmann Method, online book on North Dakota State University website

Shiyi Chen and Gary D. Doolen, Lattice Boltzmann Method For Fluid Flows, Annu. Rev. Fluid Mech. 1998. 30:329-64

http://www.openlb.org/

ROY, S., KELSO, R.M., and BAKER, A.J. (1994) ``An efficient CFD algorithm for the prediction of contaminant dispersion in room air motion", ASHRAE Transactions, 100(2), 980-987.

Jones P J, Whittle G E. Computational fluid Dynamics for building airflow prediction. Buildings and Environment, 1992, 27, 321-338.

L. B. Lucy. A numerical approach to the testing of the fission hypothesis. The Astronomical Journal, 82:1013-1024, 1977.

R. A. Gingold and J. J. Monaghan. Smoothed particle hydrodynamics: theory and application to non-spherical stars. Monthly Notices of the Royal Astronomical Society, 181:375- 398, 1977.

J. J. Monaghan. Smoothed particle hydrodynamics. Annual Review of Astronomy and Astrophysics, 30:543-574, 1992.

S. A. Munzel. Smoothed Particle Hydrodynamics und ihre Anwendung auf Akkretionsscheiben. PhD thesis, Eberhard-Karls Universitat Thubingen, 1996.

Stam J. Stable fluids. In: Proceedings of 26th international conference on computer graphics and interactive techniques, SIGGRAPH'99, Los Angeles; 1999.

Harris MJ. Real-time cloud simulation and rendering. Ph.D. thesis, University of North Carolina at Chapel Hill; 2003.

Song O-Y, Shin H, Ko H.- S. Stable but nondissipative water. ACM Transactions on Graphics 2005;24(1):81-97.

Zuo W, Chen Q. Validation of fast fluid dynamics for room airflow. In: Proceedings of the 10th international IBPSA conference, Building Simulation 2007, Beijing, China; 2007.

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

Zuo, W. and Chen, Q. 2011. "Validation of a fast-fluid-dynamics model for predicting distribution of particles with low Stokes number," Proceedings of the 12th International Conference on Indoor Air Quality and Climate (Indoor Air 2011) , Austin, Texas.

Zuo, W. and Chen, Q. 2010. "Improvements on the fast fluid dynamic model for indoor airflow simulation," Proceedings of SimBuild 2010 Conference, New York City, NY,

Zuo, W., Hu, J., and Chen, Q. 2010. "Improvements on FFD modeling by using different numerical schemes," Numerical Heat Transfer, Part B: Fundamentals, 58(1), 1-16.

Zuo, W., Jin, M., and Chen, Q. "Reduction of numerical diffusion in the FFD model,"Accepted by Engineering Applications of Computational Fluid Mechanics.

Jahangir and Jagath, 1998 M. Jahangir and J.K. Jagath, Application of artificial neural network and genetic algorithm in flow and transport simulations, Advances in Water Resources 22 (1998) (2), pp. 145-158.

S.S. Shyam, A neural network approach for non-iterative calculation of heat transfer coefficient in fluid-particle systems, Chemical Engineering and Processing 40 (2001), pp. 363-369.

Bening, T.M. Becker and A. Delgado, Initial studies of predicting flow fields with an ANN hybrid, Advances in Engineering Software 32 (2001), pp. 895-901.

R. Masini, E. Padovani, M.E. Ricotti and E. Zio, Dynamic simulation of a steam generator by neural networks, Nuclear Engineering and Design 187 (1999), pp. 197-213.

Walid and Shyam, 1998 H.S. Walid and S.S. Shyam, An artificial neural network for non-iterative calculation of the friction factor in pipeline flow, Computers and Electronics in Agriculture 21 (1998), pp. 219-228.

A.S. Kelkar, R.L. Mahajan and R.L. Sani, Real-time physiconeural solutions for MOCVD, J. Heat Transfer-Trans. ASME 118 (1996) (4), pp. 814-821.

SMAOUI Nejib, AL-ENEZI Suad, Modelling the dynamics of nonlinear partial differential equations using neural networks, Journal of computational and applied mathematics, 2004, Volume 170, page 27-58

Mai-duy Nam, Tran-cong Thanh, Numerical solution of Navier||Stokes equations using multiquadric radial basis function networks, International journal for numerical methods in fluids, 2001, Volume 37, page 65-86

Williams, P.N., The artificial neural network solution to double diffusive convection equations using spectral methods, M.S. thesis, Rice Digital Scholarship Archive

T. Kozek, T. Roska, A Double Time - Scale CNN For Solving Two-Dimensional Navier||Stokes Equations, International Journal of Circuit Theory and Applications, Volume 24 Issue 1, Pages 49-55

Nejib Smaoui, and Ali A. Garrouch, A new approach combining Karhunen-Loeve decomposition and artificial neural network for estimating tight gas sand permeability, Journal of Petroleum Science and Engineering, Volume 18, Issues 1-2, July 1997, Pages 101-112

T. Belytschko, Y. Krongauz, D. Organ, M. Fleming, and P. Krysl, Meshless methods: An Overview and Recent Developments, Computer Methods in Applied Mechanics and Engineering, Vol. 139, 3-47 (1996).

Aaron Katz, Antony Jameson, Edge-based Meshless Methods for Compressible Flow Simulations, 46th AIAA Aerospace Sciences Meeting and Exhibit, 2008.

El Zahab, Z.; Divo, E.; Kassab, A., A meshless CFD approach for evolutionary shape optimization of bypass grafts anastomoses, Inverse Problems in Science and Engineering, Volume 17, Number 3, April 2009 , pp. 411-435(25)

Zhi Shang, Application of artificial intelligence CFD based on neural network in vapor-water two-phase flow, Engineering Applications of Artificial Intelligence, Volume 18, Issue 6, September 2005, Pages 663-671