

COMPUTATIONAL FLUID DYNAMICS FOR INDOOR ENVIRONMENT MODELING: PAST, PRESENT, AND FUTURE

Qingyan (Yan) Chen[†], Zhao Zhang, and Wangda Zuo

School of Mechanical Engineering, Purdue University, 585 Purdue Mall, West Lafayette, USA

ABSTRACT

This paper gave an overview of the past and present applications of various Computational Fluid Dynamics (CFD) methods for indoor environment modeling. Typical applications used the CFD to calculate airflow, air temperature, contaminant concentrations, and turbulence in enclosed environment for studying or designing thermal comfort and indoor air quality. With simple airflow and geometry, the CFD is capable of calculating accurately mean flow parameters but less accurately turbulence parameters. For airflow in real indoor environment, it is very challenging to measure and calculate accurately the mean and turbulence flow parameters, because neither of them are free from errors. Thus, a complete validation of the CFD results by the corresponding experimental data obtained on site is extremely difficult. In the future, CFD applications for indoor environment will deal with more complicated dynamic problems and will require a method for faster than real time simulations of airflow. The Fast Fluid Dynamics (FFD) can dramatically enhance the computing speed. By running the FFD on GPUs, it is possible to perform faster than real time simulations of airflow in indoor environment.

KEYWORDS

Computational Fluid Dynamics, Indoor environment

INTRODUCTION

Buildings and other enclosed spaces, such as transportation vehicles, are important to our health and welfare because more than 90 percent of a typical American day is spent in those environments. Important parameters for indoor environments include air velocity, air temperature, turbulent levels, and contaminant concentrations of gases, solid particles, and liquid droplets. To design a healthy and comfortable indoor environment, Computational Fluid Dynamics (CFD) has been used to calculate these important parameters (Zhai 2006). This is referred as CFD modeling of indoor environment.

PAST

Most of the CFD applications for indoor environment modeling in the last century were to calculate airflow patterns and distributions of air velocity, air temperature, turbulent intensity, and gaseous contaminants. Nielsen (1976) was the first one who applied CFD for indoor environment modeling. The CFD studies in the 1970s and early 1980s were mainly for two dimensional cases. The CFD solved the Reynolds averaged Navier-Stokes equations. More specifically, the studies used eddy-viscosity models, among which the standard $k-\epsilon$ model from Launder and Spalding (1974) was most popular. The CFD applications for indoor environment modeling in the late 1980s and the 1990s extended to three dimensional cases. However, the standard $k-\epsilon$ model still retained its popularity. The Re-Normalization Group $k-\epsilon$ model also emerged as a popular model in the late 1990s.

On the other hand, Reynolds-stress models were also occasionally used for indoor environment modeling. Although these models have a solid physical background, they are mathematically complicated and numerically unstable. Thus, they need much longer computing time. Except for a few applications, such as for flows in domains with strong curvature and anisotropic flow, the improvements on the computed results are marginal. Hence, the Reynolds-stress models did not make into the mainstream of indoor environment modeling.

Chen (1995, 1996) evaluated the performance of five popular eddy-viscosity models and three Reynolds-stress models for indoor airflow. The eddy-viscosity models are the standard $k-\epsilon$ model (Launder and Spalding 1974), a low-Reynolds-number $k-\epsilon$ model (Lam and Bremhorst 1981), a two-layer $k-\epsilon$ model (Rodi 1991), a two-scale $k-\epsilon$ model (Kim and Chen 1989), and an RNG $k-\epsilon$ model

[†] Corresponding Author: Tel: + 1 765 496 7562, Fax: + 1 765 496 7534
E-mail address: yanchen@purdue.edu

^{*}Also appeared in XXV UIT National Heat Transfer Conference, Trieste, Italy, June 18-20, 2007.

(Yakhot et al. 1992). The Reynolds stress models are two with isotropization of production assumption (RSTM-IP and RSTM-GY) (Gibson and Younis 1986, Launder et al. 1975, Malin and Younis 1990) and the other with quasi-isotropic approximation (RSTM-QI) (Launder et al. 1975). Since the basic features of indoor airflow are either buoyancy-driven or pressure-gradient-driven flows, Chen (1995, 1996) evaluated those eight turbulence models for:

- Natural convection flow in a cavity
- Forced convection flow in a cavity
- Mixed convection flow in a cavity
- Impinging jet flow

Those cases are with experimental data obtained by Cheesewright et al. (1986), Restivo (1979), Schwenke (1975), and Cooper et al. (1993), respectively. The simple geometries of these cases can eliminate some measuring errors. However, the thermo-fluid boundary conditions in these measurements were sometimes difficult to control. The equipment used was very good at that time but not comparable to the best at present. Therefore, the experimental data were not of the best quality.

Table 1 summarizes the performance of the eight turbulence models for the four cases. Chen concluded that some of the models performed better in one case but poorer in another. Both the standard and RNG $k-\epsilon$ models gave acceptable results. The RNG $k-\epsilon$ model was slightly better than the standard $k-\epsilon$ model. The three Reynolds-stress models gave very close results; and the results were not significantly better than those obtained by the standard $k-\epsilon$ model. The Reynolds-stress models used much greater computing efforts. That is why the standard and RNG $k-\epsilon$ models were most popular in most CFD modeling of indoor environment. In many applications, CFD becomes an acronym for turbulence modeling with the standard $k-\epsilon$ model.

In the last century, the CFD modeling for indoor environment was for design, evaluation, and study of thermal comfort and indoor air quality, but the cases were not very complicated due to the limitations on computer capacity, computer memory, and interface for data inputs. CFD was also used to improve the accuracy of other building simulations, such as energy analysis.

Table 1. Summary of the performance of various eddy viscosity and Reynolds stress models tested by Chen (1995, 1996)

| Cases | Items | $k-\epsilon$ | LB $k-\epsilon$ | 2L $k-\epsilon$ | 2S $k-\epsilon$ | RNG $k-\epsilon$ | RSTM -IP | RSTM -GY | RSTM -QI |
|--------------------|---------------|--------------|--------------------|--------------------|--------------------|---------------------|-------------|-------------|-------------|
| Natural convection | Mean velocity | B | A | B | B | B | B | B | B |
| | Turbulence | C | C | D | D | C | C | C | C |
| | Temperature | B | D | B | B | B | B | B | B |
| | Heat transfer | C | B | A | C | C | C | C | C |
| Forced convection | Mean velocity | C | C | C | F | C | C | C | C |
| | Turbulence | D | D | D | D | D | D | D | D |
| Mixed convection | Temperature | A | A | C | A | A | A | A | A |
| | Re-attachment | C | B | B | D | A | D | D | D |
| Impinging jet | Mean velocity | C | C | C | A | A | B | B | B |
| | Turbulence | D | D | D | C | C | D | D | D |

A = good, B = acceptable, C = marginal, D = poor, F = unacceptable

PRESENT

Due to the improvements on the turbulence modeling and computer speed and memory, the CFD applications for indoor environment model have reached to a record high level at present. CFD has been used to calculate contaminant transport in gaseous, particulate, and liquid droplet forms. The simulations often include not only convection, but also conduction in solid and radiation between surfaces and within absorbing media.

The modeling is no longer limited to Reynolds averaged Navier-Stokes equations. More advanced models, such as Large Eddy Simulation (LES), start to appear in indoor environment modeling (Emmerich and McGrattan 1998, Jiang and Chen 2001). In the last ten years, a number of new RANS models have been further developed, tested and used. The trends in selecting a suitable CFD method for indoor environment modeling at present are based on accuracy and speed.

For example, natural ventilation is being considered as a sustainable measure to ventilate buildings for improving indoor air quality and reducing energy consumption by HVAC systems. The design and study of natural ventilation would not be satisfied by the mean values of airflow. Thus, LES becomes very attractive because it provides very detailed transient flow information and turbulence structure. In many cases, it is desirable to use an eddy-viscosity model near a wall to reduce computing time required by LES. This forms the detached eddy simulation (DES).

On the other hand, it is important to simulate quickly airflow and contaminant transport. For example, to protect buildings from chemical/biological warfare agents released by a terrorist, it is important to perform faster than real time simulations of airflow and contaminant transport. Another example is for the coupled airflow and energy simulation in buildings. CFD model can greatly enhance the accuracy of energy calculation by providing air temperature distributions and convective heat transfer coefficients. The CFD simulation must not use too much computing time in order to perform energy calculation within an acceptable time frame.

Therefore, simplified turbulence models are rather popular nowadays because they provide reasonably accurate results while reduce significantly computing time required. Such simplified models include zero-equation models and models using Euler equations.

In order to evaluate different new CFD methods emerged recently, Zhang et al. (2007b) recently compared 17 turbulence models for different indoor airflows. The airflows studied are:

- Natural convection flow in a cavity (Betts and Bokhari 2000)
- Forced convection flow in a cavity (Ito et al. 2000)
- Mixed convection flow in a cavity (Blay et al. 1992)
- Flow with very strong buoyancy in a room (Murakami et al. 1995)

Note that the experimental data for these cases are of much higher quality than those used by Chen (1995, 1996). The corresponding thermo-fluid boundary conditions were very well controlled. The 17 models tested range widely from zero-equation models to DES. Table 2 compares the best model from each category: the indoor zero-equation model (0-eq.) by Chen and Xu (1998), the RNG $k-\epsilon$ model by Yakhot et al. (1992), the SST $k-\omega$ model (SST) by Menter (1994), a low Reynolds number $k-\epsilon$ model (LRN-LS) by Launder and Sharma (1974), a modified $v2f$ model ($v2f-dav$) by Davidson et al. (2003), a Reynolds stress model (RSM-IP) by Launder et al. (1975), a LES with dynamic subgrid scale (SGS) model (Germano et al. 1991, Lilly 1992), and a DES by Shur et al. (1999).

Table 2. Summary of the performance of various turbulence models tested by Zhang et al. (2007b)

| Cases | Compared items | Turbulence models | | | | | | | |
|-----------------------|------------------|-------------------|---------------------|-------------------|------------|-------------|------------|---------------|-----|
| | | 0-eq. | RNG $k-\epsilon$ | SST $k-\omega$ | LRN -LS | V2f -dav | RSM -IP | DES | LES |
| Natural convection | Mean temperature | B | A | A | C | A | A | - | A |
| | Mean Velocity | D | B | A | B | A | B | - | B |
| | Turbulence | n/a | C | C | C | A | C | - | A |
| Forced convection | Mean Velocity | C | A | C | A | A | B | - | A |
| | Turbulence | n/a | B | C | B | B | B | - | B |
| Mixed convection | Mean temperature | A | A | A | A | A | B | B | A |
| | Mean Velocity | A | B | B | B | A | A | B | B |
| | Turbulence | n/a | A | D | B | A | A | B | B |
| Strong buoyancy flow | Mean temperature | A | A | A | A | A | n/c | - | B |
| | Mean Velocity | B | A | A | A | A | n/c | - | A |
| | Turbulence | n/a | C | A | B | B | n/c | - | B |
| Computing time (unit) | | 1 | 2 - 4 | | 4 - 8 | | 10 - 20 | $10^2 - 10^3$ | |

Not all the models tested were developed recently, but they were selected because of their popularity or potential in indoor environment modeling. The overall performance of the RNG $k-\epsilon$ model is still one of the best. Although the results of the standard $k-\epsilon$ model are not shown here, they are very close to those of the RNG $k-\epsilon$ model. The $v2f-dav$ model has better accuracy than the RNG $k-\epsilon$ model but requires slightly longer computing time. Again the Reynolds-stress model has problem in obtaining a converged solution. The LES and DES provide the most detailed information of airflow while the computing time is much higher than the other models and their accuracy may not always be the best. The indoor zero equation model is least accurate, but it converges very fast.

The evaluation does not recommend a single model for indoor environment modeling. The selection of a model depends on the applications. Clearly, for studying the mechanism of natural ventilation, LES and DES are the best. For fast prediction of airflow and contaminant transport, the indoor zero-equation model should be used. For general indoor environment design and studies, the RNG $k-\epsilon$ model seems still appropriate while the modified $v2f$ model can be promising.

Both Tables 1 and 2 show that the CFD models are capable in predicting indoor airflow, because the differences between the computed results and measured data are generally less than 30% for the mean parameters. However, in engineering practice, comparisons of CFD results with experimental data measured on site often show much larger discrepancies. Why the CFD methods perform reasonably for

the flows shown in the two tables and badly for flows in reality? Due to its performance for flows in reality, CFD has also earned another acronym – Colored Fluid Dynamics. Why does nobody believe your results except yourself if you use CFD, and why does everyone believe your results except yourself if you measure fluid flow?

To analyze the problem, this study used the CFD with the RNG k- ϵ model to calculate airflow and the distributions of air temperature, a gaseous contaminant, and a particulate contaminant in an enclosed environment – a section of a twin-aisle airliner cabin mockup (Zhang et al. 2007a) as shown in Fig. 1.

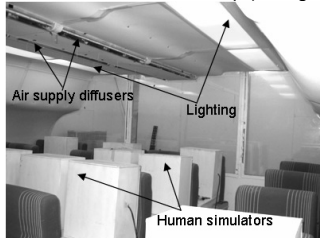


Figure 1. A view of the twin-aisle airliner cabin mockup

Figure 2 shows that the cabin mockup had four rows with 28 seats. A half of the seats were occupied by heated human simulators. Conditioned air was supplied from the linear slot diffusers from the ceiling level and the polluted air was extracted from outlets on the sidewalls near the floor. The airflow rate supplied was close to 10 L/s per seat. A tracer gas (SF_6) was used to simulate a gaseous contaminant and non-evaporative, monodispersed particles were generated to simulate a particulate contaminant. Both the contaminants were released at the top of a human simulator as shown in Fig. 2.

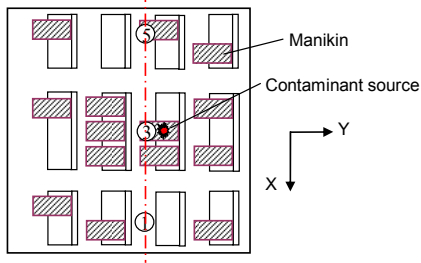


Figure 2. The plane view of the cabin mockup and measurement positions

Figure 3 compares the computed airflow pattern with the measured one. The experiment measured fairly accurate airflow pattern by using ultrasonic anemometers. However, the airflow from the inlet was not accurately measured, because the head of the ultrasonic anemometers was too bulky. Instead, omni-directional hot-sphere anemometers were used with smoke visualization, which have great uncertainties. The measured data from the inlet diffusers were then used as the boundary conditions in the CFD simulation. Due to the uncertainties in the boundary conditions, the airflow pattern computed do not agree with the measured one. In this case, both the experimental measurements and the turbulence model contribute to the discrepancies. Often one would trust the measured data not the computed results that may not be fair for this case.

Figure 4 further compares the distributions of air temperature and SF_6 and particles concentrations in different locations in the cabin mockup. Although the air was very well mixed in the cabin, the concentration distributions were not uniform due to the concentrated contaminant sources. Generally, the CFD could predict reasonably the profiles but the discrepancies can be more than 100%. Again, both the experiment measurements and the CFD simulations were not free from errors. In addition, the flow in the cabin was highly turbulent. The turbulence intensity can be one magnitude order higher than the mean air velocity. With the large discrepancies on the airflow pattern, it is not realistic to expect a good

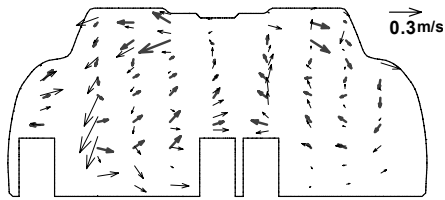


Figure 3. Comparison of the computed and measured airflow pattern in a cross section of the cabin mockup

agreement between the computed and measured air temperature and contaminant concentrations. The cabin mockup was with controlled boundary conditions so that the flow was stable and the surrounding temperature did not change. If the experimental data were obtained on site where one has little control on the boundary conditions, a satisfactory validation of the computed results would be almost impossible. Even if a good agreement were achieved, the results could not be trusted.

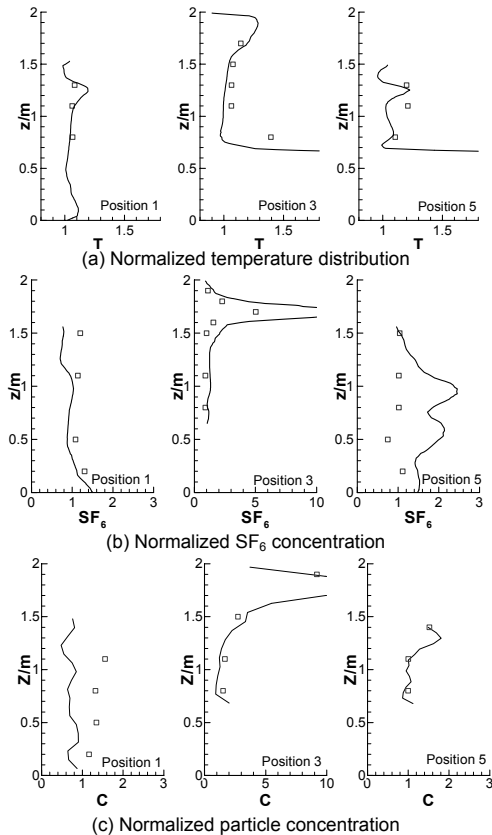


Figure 4. Comparison of the computed and measured temperature and contaminant concentration distributions. All quantities are normalized by $\phi = (\phi - \phi_{in}) / (\phi_{out} - \phi_{in})$.

Nevertheless, indoor environment design often uses CFD to optimize or identify the most influential parameters. Although CFD simulations may not give accurate results, they can accurately estimate the trend due to the change of the parameters. Then the CFD is very useful. Since the CFD is much cheaper than experimental measurements, CFD has become a popular tool for indoor environment design and modeling.

FUTURE

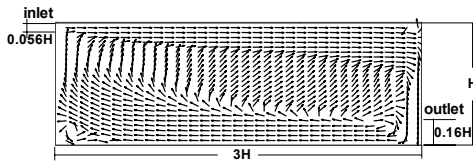
No doubt, building engineers and scientists will continue to seek more accurate CFD methods for indoor environment modeling in the future. The development on CFD methods in the last 30 years did not find a universal model for indoor airflow. The accuracy of the current CFD methods, such as the RNG $k-\epsilon$ model and the $v2f$ -dav model, may be sufficient for most applications in indoor environment modeling.

However, current CFD methods are mainly used for relatively simple applications under steady state conditions. Indoor environment is for its occupants. Most of the occupants would move around and the movement would create a huge impact on thermal comfort and indoor air quality. The simulations of moving body in indoor environment by CFD are challenging. Recently, Mazumdar and Chen (2007) used successfully combined static and dynamic meshes to simulate a moving crew in an airliner cabin. The crew member was simulated by a rectangular block. The results look very interesting and promising (Mazumdar and Chen 2007).

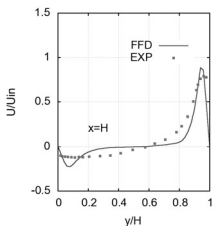
If the simulation is extended to the movement of the arms and legs of the crew, it will add a lot of complexities. This is one of the many dynamic applications that the next generation of CFD methods should face.

Furthermore, the CFD methods with simple turbulence models are still not sufficiently fast to meet the real world challenges in predicting contaminant transport in indoor environment. As discussed above, it is desirable to perform faster than real time prediction of contaminant transport in indoor environment in case of a chemical/biological leakage or terrorist attack. None of the CFD methods discussed above can give real time simulations of the contaminant transport even for a space as small as an individual office.

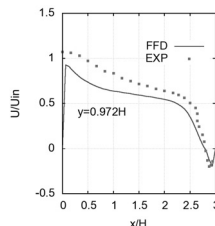
On the other hand, weather forecast has realized faster than real time simulations of atmosphere motion and temperature. The simulations use a semi-Lagrangian scheme (Robert et al. 1972) to increase the time step size at little additional cost and without degrading the accuracy of the solution. Stam (1999) conducted pioneer work by using this method for fluid motion in computer games and achieved plausible results. Since the semi-Lagrangian approach has been successfully used in weather forecast and computer games, one might use this method in the future to predict indoor airflow on real time or even faster than real time. This paper calls such an approach Fast Fluid Dynamics (FFD).



(a) The velocity field predicted by FFD



(b) Comparison of the mean horizontal velocities by the FFD with the data (Restivo 1979) at vertical line $x=H$



(c) Comparison of the mean horizontal velocities by the FFD with the data (Restivo 1979) at horizontal line $y=0.972H$

Figure 5. Airflow in a ventilated room predicted by FFD

Zuo and Chen (2007) used the FFD to simulate (1) flow in a lid-driven cavity, (2) flow in a plane channel, and (3) flow in a ventilated room. By comparing the computed results with corresponding data from the

literature, the FFD can predict such flows with reasonable accuracy and the simulations were faster than real time. Figure 5 shows the results calculated by FFD for the airflow in a ventilated room and the comparison of the computed velocity profiles in sections $x/H = 1$ and $y/H = 0.972$ with the measured data. The total grid number used was 300×125 . The simulation was 2.4 times faster than real time with a 0.5 s time step.

Currently, the FFD is able to simulate the airflow on real-time in a building with 100,000 grids. In addition, the recent rapid increase in the speed and programmability of graphics processors units (GPU) makes it possible to use GPUs for scientific computation as demonstrated by Harris (2003). As illustrated in Figure 6, GPU performance has increased at a much faster rate than CPUs. Liu et al. (2004) showed that GPU is 14 times faster than CPU for FFD simulations. The simulation in CPU is performed on a grid of cells. Programmable GPUs are optimized for performing computations on pixels, which is analogue to a grid of cells. GPUs achieve high performance through parallelism, because they are capable of processing multiple vertices and pixels simultaneously. To develop a FFD program for GPU computation would make faster than real time simulation possible for contaminant transport in a reasonable size of buildings in the near future.

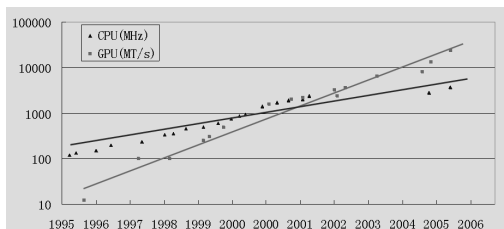


Figure 6. The hardware performance increasing of GPU (NAVIDA) and CPU (INTEL) over past ten years

CONCLUDING REMARKS

This paper gave an overview of using CFD for indoor environment modeling and led to the following concluding remarks:

The applications in the 1980s and 1990s were relatively simple. The CFD models at that time were mainly eddy-viscosity models that can provide reasonably accurate results for the mean parameters, such as air velocity, air temperature, and contaminant concentrations.

The applications at present deal with more realistic indoor environment, although most of the studies were for steady state conditions. In addition to seeking more accurate models, fast simulations are a trend. The CFD models used today range from eddy-viscosity, Reynolds-stress, large-eddy-simulation, and detached-eddy-simulation ones. The accuracy of those models does not improve very much compared with that used in the 1980s and 1990s. However, the models can give more information and can be much faster. If airflow and geometry are complex, it is very challenging to measure and calculate accurately the mean and turbulence parameters. Thus, a complete validation of CFD by the corresponding experimental data measured on site is extremely difficult.

The applications in the future could include dynamic boundary conditions, such as moving body in indoor environment. At the same time, research in indoor environment modeling will continue to seek fast CFD models with reasonable accuracy. The FFD method can dramatically reduce the computing speed. It is possible to perform fast than real time simulations by FFD on GPU.

ACKNOWLEDGMENT

The authors are grateful to the contributions made by Dr. Wei Zhang of Boeing Commercial Group and by Xi Chen, Sagnik Mazumdar, and Tengfei Zhang from Purdue University. This research presented in this paper was partially funded by the 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 the research.

REFERENCES

1. P. L. Betts and I. H. Bokhari (2000) "Experiments on turbulent natural convection in an enclosed tall cavity", *International Journal of Heat and Fluid Flow*, Vol. 21, 675-683.
2. D. Blay, S. Mergui and C. Niculae (1992) "Confined turbulent mixed convection in the presence of horizontal buoyant wall jet", *Fundamentals of Mixed Convection*, Vol. 213, 65-72
3. R. Cheesewright, K. J. King and S. Ziai (1986) "Experimental data for the validation of computer codes for the prediction of two-dimensional buoyant cavity flows". Significant questions in buoyancy affected enclosure or cavity flows, J. A. C. Humphrey, C. T. Adedisian and B. W. L. Tournau (Eds.), 75-81, ASME.
4. Q. Chen (1995) "Comparison of different $k-\epsilon$ models for indoor air-flow computations", *Numerical Heat Transfer Part B: Fundamentals*, Vol. 28, 353-369.
5. Q. Chen (1996) "Prediction of room air motion by Reynolds-stress models", *Building and Environment*, Vol. 31, 233-244.
6. Q. Y. Chen and W. R. Xu (1998) "A zero-equation turbulence model for indoor airflow simulation", *Energy and Buildings*, Vol. 28, 137-144.
7. D. Cooper, D. C. Jackson, B. E. Launder and G. X. Liao (1993) "Impinging jet studies for turbulence model assessment - I: Flow-field experiments", *International Journal of Heat and Mass Transfer*, Vol. 36, 2675-2684.
8. L. Davidson, P. V. Nielsen and A. Sveningsson (2003) "Modification of the $v2f$ model for computing the flow in a 3D wall jet", *Turbulence Heat and Mass Transfer*, Vol. 4, 577-584.
9. S. J. Emmerich and K. B. Mcgrattan (1998) "Application of a large eddy simulation model to study room airflow", *ASHRAE Transactions*, Vol. 104, 1128-1140.
10. M. Germano, U. Piomelli, P. Moin and W. H. Cabot (1991) "A dynamic subgrid-scale eddy viscosity model", *Physics of Fluids A: Fluid Dynamics*, Vol. 3, 1760-1765.
11. M. M. Gibson and B. A. Younis (1986) "Calculation of swirling jets with a Reynolds stress closure", *Physics of Fluids*, Vol. 29, 38-48.
12. M. J. Harris (2003) "Real-time cloud simulation and rendering", Ph.D. Thesis, University of North Carolina at Chapel Hill.
13. K. Ito, S. Kato and S. Murakami (2000) "Model experiment of flow and temperature field in room for validating numerical simulation analysis of newly proposed ventilation effectiveness", *J. Archit. Plann. Environ. Eng. (In Japanese)*, Vol. 534, 49-56.
14. Y. Jiang and Q. Y. Chen (2001) "Study of natural ventilation in buildings by large eddy simulation", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 89, 1155-1178.
15. S. W. Kim and C. P. Chen (1989) "A multiple-time-scale turbulence model based on variable partitioning of the turbulent kinetic-energy spectrum", *Numerical Heat Transfer Part B: Fundamentals*, Vol. 16, 193-211.
16. C. K. G. Lam and K. Bremhorst (1981) "A modified form of the $k-\epsilon$ model for predicting wall turbulence", *Journal of Fluids Engineering-Transactions of the ASME*, Vol. 103, 456-460.
17. B. E. Launder and B. I. Sharma (1974) "Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc", *Letters in Heat Mass Transfer*, Vol. 1, 131-138.
18. B. E. Launder and D. B. Spalding (1974) "The numerical computation of turbulent flows", *Computer Methods in Applied Mechanics and Engineering* Vol. 3, 269-289.
19. H. Launder, P. Moin and J. Kim (1975) "Progress in the development of a Reynolds-stress turbulence closure", *J. Fluid Mechanics*, Vol. 68, 537-566.
20. D. K. Lilly (1992) "A proposed modification of the germano subgrid-scale closure model", *Physics of Fluids A: Fluid Dynamics*, Vol. 4, 633-635.
21. Y. Liu, X. Liu and E. Wu (2004) "Real-time 3D fluid simulation on GPU with complex obstacles", *Proc. of 12th Pacific Conference on Computer and Applications (PG'04)*.
22. M. R. Malin and B. A. Younis (1990) "Calculation of turbulent buoyant plumes with a Reynolds stress and heat-flux transport closure", *International Journal of Heat and Mass Transfer*, Vol. 33, 2247-2264.
23. S. Mazumdar and Q. Chen (2007) "Impact of moving bodies on airflow and contaminant transport inside aircraft cabins", *Proc. of ROOMVENT 2007, Helsinki, Finland*.
24. F. R. Menter (1994) "Two-equation eddy-viscosity turbulence models for engineering applications", *AIAA Journal*, Vol. 32, 1598-1605.
25. S. Murakami, S. Kato and R. Yoshie (1995) "Measurement of turbulence statistics in a model fire room by LDV", *ASHRAE Transactions*, Vol. 101, 3905.
26. P. V. Nielsen (1976) "Flow in air conditioned rooms (english translation, 1974)", Ph.D. Thesis, Technical University of Denmark.
27. A. Restivo (1979) "Turbulent flow in ventilated room", Ph.D. Thesis, University of London (U.K.).

28. A. Robert, C. Turnbull and J. Henderso (1972) "Implicit time integration scheme for baroclinic models of atmosphere", *Monthly Weather Review*, Vol. 100, 329-335.
29. W. Rodi (1991) "Experience with two-layer models combining the k-epsilon model with a one-equation model near the wall", *Proc. of 29th Aerospace Sciences Meeting*, Nevada.
30. H. Schwenke (1975) "Über das verhalten ebener horizontaler zuluftstrahlen im begrenzten raum", *Luft- und Kältetechnik*, Vol. 5, 241-246.
31. M. Shur, P. R. Spalart, M. Strelets and A. Travin. (1999). "Detached-eddy simulation of an airfoil at high angle of attack," In 4th Int. Symposium on Eng. Turb. Modeling and Experiments.
32. J. Stam (1999) "Stable fluids", *Proc. of SIGGRAPH'99*.
33. V. Yakhot, S. A. Orszag, S. Thangam, T. B. Gatski and C. G. Speziale (1992) "Development of turbulence models for shear flows by a double expansion technique", *Physics of Fluids A: Fluid Dynamics*, Vol. 4, 1510-1520.
34. Z. Zhai (2006) "Application of computational fluid dynamics in building design: Aspects and trends", *Indoor and Built Environment*, Vol. 15, 305-313.
35. Z. Zhang, X. Chen, S. Mazumdar, T. Zhang and Q. Chen (2007a) "Experimental and numerical investigation of airflow and contaminant transport in an airliner cabin mock-up", *Proc. of ROOMVENT 2007*, Helsinki, Finland.
36. Z. Zhang, J. Z. Zhai and Q. Chen (2007b) "Evaluation of various CFD models in predicting room airflow and turbulence", *Proc. of ROOMVENT 2007*, Helsinki, Finland.
37. W. Zuo and Q. Chen (2007) "Validation of fast fluid dynamics for room airflow", *Proc. of International Symposium on Heating, Ventilating and Air Conditioning*, Beijing, China.