12th REHVA World Congress

CLIMA2016

The Status of Computational Fluid Dynamics

By: Peter V. Nielsen[1], Li Liu[1], Lei Peng[2], Yuguo Li[2]

[1] Department of Civil Engineering, Aalborg University, Aalborg, Denmark
[2] Department of Mechanical Engineering, The University of Hong Kong, Hong Kong SAR, China




Peter V. Nielsen. *Li Liu.* *Lei Peng.* *Yuguo Li.*

Summary

Computational fluid dynamics (CFD) was first introduced in the building ventilation industry in the 1970s. Since then, it has been increasingly used as testified by the growth of the number of peer-reviewed articles, which was less than 10 per year in the 1990s and 60 to 70 per year in the recent years. The CFD market for all different types of fluid flow indicates now a level of at least USD 700 million per year with an annual growth rate of 13%.

This article presents a short discussion of the development of CFD by showing two examples: an early case and a recent study on the cross-infection risks from a person's exhalation of particles. The article will also address the use of benchmark tests as a quality control of CFD.

Introduction

The indoor environment community has adopted computational fluid dynamics (CFD) as a useful tool for the prediction of air movement in ventilated spaces. The method has been used for more than 40 years as a research tool [1 and 2]. Now it is used routinely in civil engineering when designing a large or complicated air distribution system [3].

The airflow in a room – and flow in general – is described mathematically by a set of coupled nonlinear differential equations, which is known as the Navier-Stokes equations. These equations can be solved analytically only for simple and ideal conditions. For the general flow e.g. in a ventilated room, it is necessary to divide the room space into a high number of small cells and reformulate the differential equations into a high number of ordinary equations, distributed in the cells, and solve those equations by a numerical method. This is called Computational Fluid Dynamics.

The development of CFD models for room air movement is strongly influenced by the increased computer power that has been available for the past few decades. This development includes an increasing computer speed, but also a decreasing cost corresponding to a reduction by a factor of 10 every eighth years.

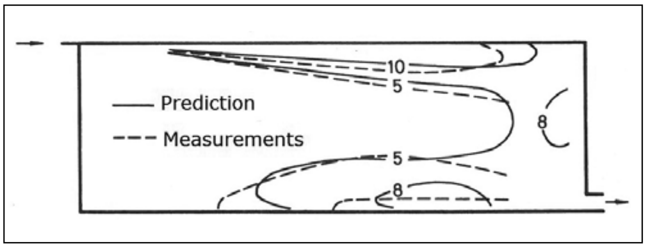


Figure 1. Measurement and prediction of the velocity distribution (percentage of supply velocity) in a mixing ventilated room.

An early CFD prediction

A CFD prediction from 1973 shows the isothermal two-dimensional velocity distribution in a room ventilated by a full-width slot, as shown in Figure 1 [1]. Due to the limited computer power, a steady two-dimensional isothermal flow was the only possibility for a CFD simulation in the beginning of the 1970s. The number of equations was even reduced because a special set of equations were used instead of the original continuity equation and the three Navier-Stokes equations. The turbulence effect was described by the so-called standard k-ε model.

The prediction was based on 7×10 cells, and the computer had a mainframe memory of only 158 kB in the initial phase of this work! The use of the so-called box method and the use of cells with large differences in size gave promising results, but the distribution of cell size requires some information on the flow before the prediction is made. Despite of all these constraints, the agreement of the predicted air flow pattern with the measured one was reasonably good.

An example of present day CFD application

The CFD method of today shows a lot of possibilities within air distribution in rooms. This could for example be design with complicated furnishing of an operation theatre, models for movement of people, design of an air supply opening, smoke management in buildings and underground stations, human body micro-environment and cross infection risks [3].

Figure 2 shows an example where CFD simulations are used to predict the transport of droplet nuclei or large droplets

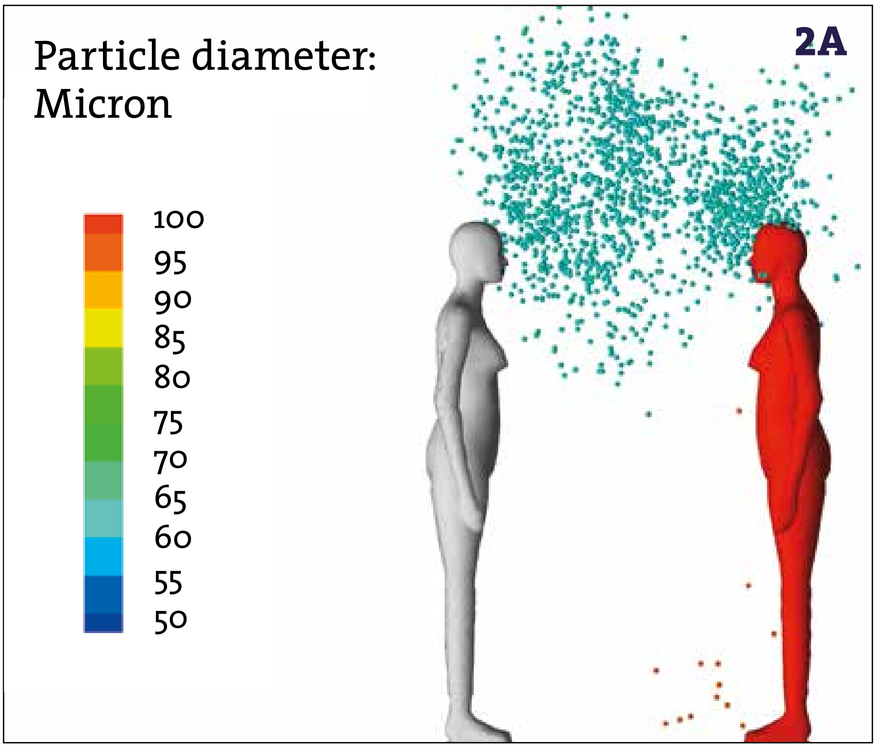
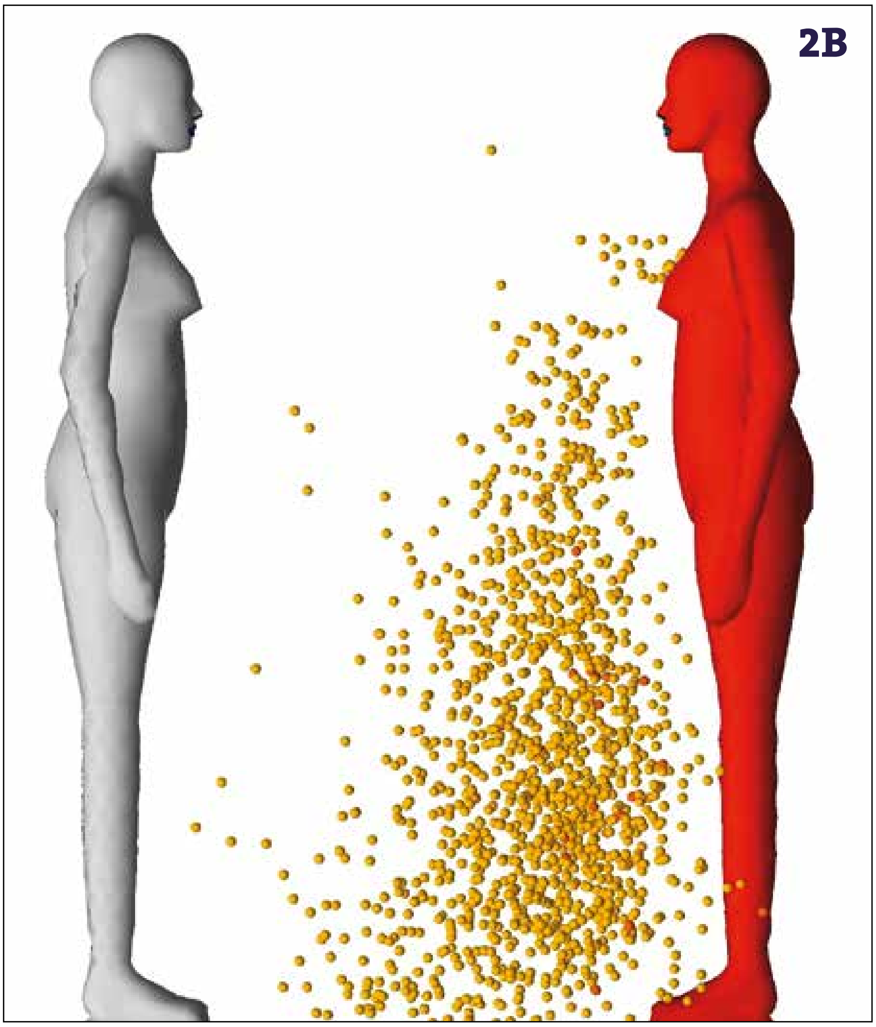


Figure 2A shows the exhalation of 100 µm droplets at 35% RH, and Figure 2B shows the same exhalation to the surrounding air at a 95% RH both after 12 sec of exhalation. The evaporation of the droplets let them rise to the upper part of the room in case of 35 % RH as droplet nuclei, as shown on Figure 2A.



between people [4]. The reference shows a CFD model where a transition can take place from droplet-borne infection to airborne infection, because the exhaled droplets can evaporate in the air and transform into droplet nuclei, depending on the humidity in the surroundings of the exhalation flow.

The two predictions in Figure 1 and Figure 2 show the significant advancement that has been taking place in the last 40 years, cf. Table 1. First, a much more complicated flow can be solved today, including e.g. evaporating droplets in a time-dependent flow. The grid has been developed from simple rectangular cells to body-fitted tetrahedron cells. Blocks with different types of cells can be used and different grid adaptations as e.g. velocity or gradient adaption are also available. The numerical scheme has been developed from the first order to third order of accuracy in many present predictions. Development of new turbulence models has been taking place, and the number of grids has been increased by a factor of 10^5.

It is the development in computer speed and size which enable the large number of cells. Table 1 shows how the computer speed has increased by a factor of 10^3 and the capacity by a factor of 10^4, and super computers today do naturally show much larger capacity.

The turbulence models

The air movement in a ventilated room is turbulent. In practice, it is impossible to make a direct numerical simulation of this flow even though it is fully described by the Navier-Stokes equations, because it requires an extremely high number of cells to describe the turbulence in the fluctuating flow. It is possible to work with a practical level of cell numbers if the numerical simulation is based on averaged variables. The fluctuating flow exhibits an apparent increase in resistance to deformation, and this effect is expressed as an extra (turbulence) viscosity which can be solved by a turbulence model.

A number of turbulence models have been developed during the years and some of those, but not all models, are relevant for room air motion. In the ▶

1970s, the Spalding group at Imperial College explored the so-called *k, k–l, k–kl, k-ε* and *k–ω* models, and they realised the advantages of a "standard" model and the order that it would bring to future research and engineering use, and so they preferred the *k-ε* model, [5]. Recently Zhang et al. [6] made an overview of the usual models for the prediction of room air movement.

Figure 3 shows the outcome of a workshop in which a total of 19 international teams made predictions of the isothermal flow in a deep room with slot inlet (back-facing step flow). They used 15 different turbulence models, 2D and 3D geometry and different schemes and software, [7]. There are no measurements for a comparison with the predictions. The penetration length of the inlet jet is shown versus the inlet velocity (here given as the Reynolds number). The high velocity part of the transient area and the fully developed turbulent area are relevant for room air flow.

It is known – and seen – that all turbulence models are inaccurate in the transient area, and it is also seen that the different models give different results in the fully developed area although they all describe the same tendency. Furthermore, the results are more diverse than expected. The predicted penetration length not only differs due to the use of different codes, turbulence models, numerical schemes and convergent criteria, but also from different users even when the same model is used.

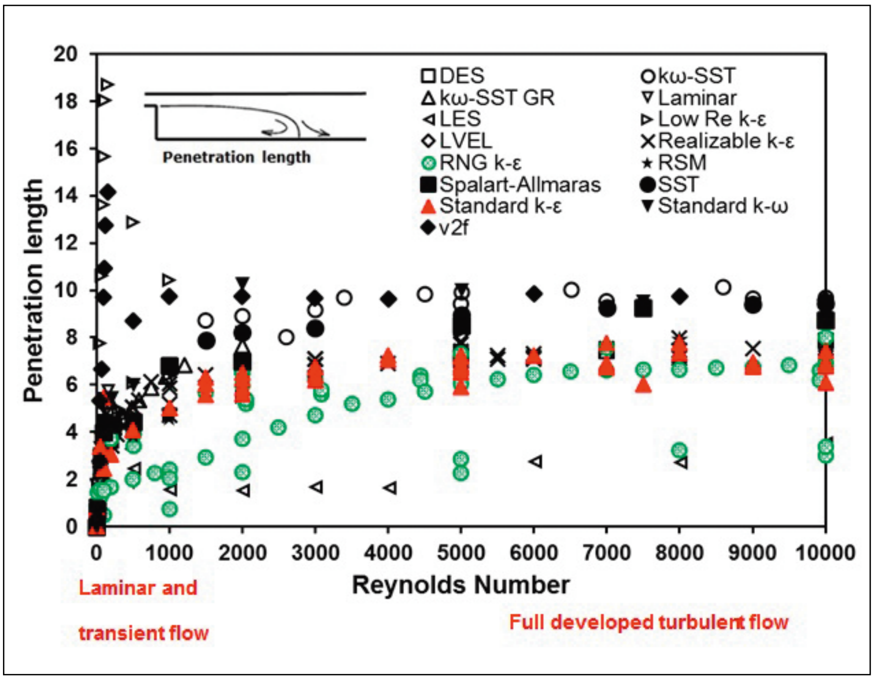


Figure 3. Predicted penetration length in a deep room versus the Reynolds number of a total of 353 data sets submitted by 19 teams.

A way to increase the quality of CFD predictions in ventilation is to test software and turbulence models in benchmarks, which is assumed to be close to the problem which has to be studied. This method is already used in connection with smoke management predictions. Figure 4 shows an example on such a procedure.

The figure shows predictions with four different turbulence models in the IEA Annex 20 Benchmark test (www.cfd-benchmarks.com). The figure indicates some differences, especially in two of the corners [8]. The grid is identical in all four cases, which means that the difference in the flow can be ascribed to the turbulence models used to predict the two-dimensional flow.

Comparisons with measurements show that both the *k-ε* and the *k-ω* model in Figure 4 perform well with the smallest deviations for the *k-ε* model. Zhang et al. [6] also show that the *RNG k-ε* model was good to acceptable for CFD predictions of forced convection. Based on the benchmark tests, it could be concluded that a *k-ε* or a *RNG k-ε* model is a promising turbulence model for prediction of isothermal 2D mixing flow as the type of flow tested in Figure 3. The two models are shown with red and green in

Predictions in	**Figure 1, 1973**	**Figure 2, 2012**
Type of flow	2D, isothermal and steady air flow	3D, non-isothermal and time dependent air flow. Evaporation of liquid droplets
Grid	Rectangular grid	Body fitted tetrahedron grids
Order of accuracy of the numerical scheme	First order	Second order
Turbulence model	*k-ε* model	*RNG k-ε* model
Number of cells	70	About 1 million
Computer facility	IBM 360 model 40	Dell Precision T1600
CPU base frequency	<100MHz	3.5 GHz
RAM	128KB	16GB

Table 1. Comparison of the CFD predictions made in 1973 and 2012.

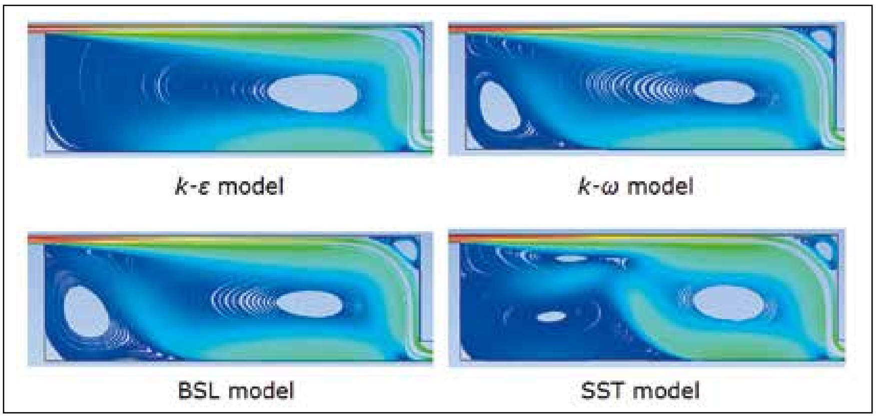


Figure 4. Two-dimensional isothermal and steady state simulations of the Annex 20 2D benchmark test. Four different turbulence models are tested.

the figure. They represent a less diverse distribution of results, but they probably also express user dependent influence.

Conclusion

Computational Fluid Dynamics has been increasingly used in the ventilation industry in the last decade. The continuing improvement of computer speed and capacity has supported this tendency, and development in the numerical schemes support the improved use of the method.

A number of turbulence models have been developed for different types of flow. Benchmark tests, or other experience, should be used to do the right selection of a model and, therefore, improve quality of predictions in a given geometry with a given air distribution.

References

[1] Nielsen PV (1973) Berechnung der Luftbewegung in einem zwangsbelüfteten Raum, *Gesundheits-Ingenieur,* 94, pp. 299-302.

[2] Jones PJ and Whittle GE (1992) Computational Fluid Dynamics for Building Air Flow Prediction – Current Status and Capabilities. *Building and Environment*. Vol. 27, No. 3, pp. 321-338.

[3] Nielsen, PV (2015), Fifty years of CFD for room air distribution, Building and Environment, vol 91, September, pp. 78–90, 10.1016/j.buildenv.2015.02.035.

[4] Liu L and Li Y (2012) Simulation of Interpersonal Transport of Expiratory Droplets and Droplet Nuclei between Two Standing Manikins. in Healthy Buildings 2012, 10th International Conference. Queensland University of Technology, Brisbane, Australia.

[5] Runchal AK (2009) Brian Spalding: CFD and reality – A personal recollection, International *Journal of Heat and Mass Transfer,* 52, pp. 4063–4073.

[6] Zhang Z, Zhang W, Zhai Z and Chen Q (2007) Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part-2: comparison with experimental data from literature, *HVAC&R Research*, 13(6).

[7] Peng L, Nielsen PV, Wang X, Sadrizadeh S, Liu L & Li Y (2016), Possible User-Dependent CFD Predictions of Transitional Flow in Building Ventilation, Building and Environment, vol 99, no. April., 10.1016/j.buildenv.2016.01.014

[8] Nielsen PV, Rong L and Olmedo I (2010) The IEA Annex 20 Two-Dimensional Benchmark Test for CFD Predictions, ISBN 978-975-6907-14-6, Clima 2010, 10th REHVA World Congress.