Hot Issues in Fire Engineering

28 March 2013

Mistakes Made in the Application of Computational Fluid Dynamics

W.K. Chow
Fellow, Hong Kong Academy of Engineering Sciences
Research Centre for Fire Engineering, Department of Building Services Engineering
The Hong Kong Polytechnic University, Hong Kong, China

1. Introduction

Numerical Heat Transfer and Fluid Flow, or sometimes called Computational Fluid Dynamics (CFD), has been used [1,2] in fire hazard assessment in performance-based design (PBD) or Fire Engineering Approach (FEA) over the past three decades. Even from day one, there are numerous problems because two-dimensional simulations are conducted with only 100,000 cells. It is well-known that only averaged flow parameters can be predicted. However, the prediction was used to 'tune' up adjustable parameters of fire hazards, such as estimating Available Safe Egress Time (ASET) [3] based on CFD predictions with unknown certainties.

Most of the CFD models are not properly validated due to high cost and other difficulties in conducting large-scale fire tests. Consequently, performance-based designs using those models are always challenged by different parties. The free CFD software Fire Dynamics Simulator FDS [4] – which is developed by the National Institute of Standards and Technology in USA – is now a common design tool. In Hong Kong, hot smoke tests are required [5,6] in halls of irregular shape or buildings taller than 12 m in order to evaluate the performance of smoke exhaust systems. Therefore, CFD predictions must be used carefully in new PBD projects. Previous projects with fire safety provisions deduced from CFD results must be reviewed for upgrades, the software fire safety management at the very least.

The accuracy of predictions depends on many parameters. There are at least three groups of parameters. The first group includes parameters in physical models, such as the empirical coefficients C_1, C_2, C_D, σ_R, σ_ε, μ_t,…, etc. in Reynolds-averaged Navier-Stokes (RANS) equations [7]; and Smagorinsky constant C_S in Large Eddy Simulation (LES) turbulent models [8]. Most of the physical parameters are measured by bench-scale experiments. Tuning up the parameters in building fire simulations can be criticized as a curve-fitting exercise. The second group is numerical parameters such as discretization, spatial and temporal parameters, and convergence criteria. They can be derived mathematically from experiments of larger scales, such as on thermal plumes [9]. The third group is parameters on the simulation of physical systems. The boundary condition of the building is an obvious example. There are no rules in determining such parameters, for example, the extension of the computing domain outside an opening. All values are derived from experience.

2. Common Mistakes

Free CFD software is commonly used and there is no guarantee that the software is appropriate for simulating big building fires. Below are the problems identified [10,11] in the application of Computational Fluid Dynamics in the Far East:

- Air pressure and turbulence parameters are seldom presented, and only the velocity vector patterns and temperature distribution are shown.
- Grid sensitivity criteria are only deduced from temperature and velocity predictions, but not pressure or turbulence parameters.
- There is no justification on fire phenomena by scale-models or full-scale burning tests.

Reviewers have put forth the following criticisms while reviewing journal papers where CFD was applied in fire simulations:

- All phenomena are substantially three-dimensional; the flow and temperature fields of the smoke exhaust in the hall are very unstable.
- The three-dimensionality and the instability of the flow fields are not fully discussed.
- Ability of CFD to resolve the flow in the turbulent fire plume is in doubt.
- Uncertainty in the ability of CFD to resolve the turbulent exchange flow across the opening.

More caution is needed in the application of CFD to simulate fire. CFD might only be feasible for practical smoke control design. Full-scale burning tests on typical scenarios with similar conditions are still necessary. Hot smoke tests must be carried out during the testing and commissioning of smoke exhaust system in tall halls.

Very little validation works had been done, and some good and bad examples are reported in the literature [12], as shown in Fig. 1 and Fig. 2.

Therefore, the ‘stability criteria’ and ‘free boundary conditions’ must be inspected in CFD reports at the very least.

3. Stability Criteria

Stability criteria should be monitored [9,13]:

- Grid size that is denoted by δx, δy and δz is the most important numerical parameter in CFD simulations.
- The quality of the mesh is assessed by a non-dimensional parameter, rather than an absolute mesh cell size.
- For simulations involving buoyant plumes, the performance of the flow field is measured by the non-dimensional expression on R^* in terms of a characteristic of fire diameter.

Hill et al. [14] presented a validation study which suggests that R^* should range from 0.06 to 0.25. Ma and Quintiere [9] also found that the optimal resolution of a pool fire simulation R^* is around 0.05; and they predicted the centerline temperature and velocity in the non-combusting region accurately.

Such study focused on comparing velocity and temperature, but they have not compared pressure and turbulence. Note that no data supported by full-scale burning tests on fire-induced turbulence is available. For example, a question is raised:

How can a 10 m grid include turbulence in a building to study the combustion process without using any models?

No one has come up with the appropriate grid size that can give accurate predictions on velocity and temperature, pressure and turbulence for predicting flow with realistic combustion including key intermediate reactions.

4. Free Boundary Conditions

There are always windows and doors in a building. Bi-directional flow would be observed experimentally, with hot gas flowing out and cool air coming into the room. The boundary conditions of flow parameters, pressure in particular, have to be specified carefully. There are empirical correlations among the pressure profiles across the vertical openings such as windows or doors, and room geometries, heat release rates of the fires, and opening sizes. However, such boundary conditions might not give proper specification.

A better approach is to extend the computing domain outside, as pointed out years ago by Galea and associates [15]. An example [16] is shown in Fig. 3.

5. Recommend Evaluation Guide

There is no doubt that CFD predictions should be evaluated properly with technical guidance [17]. However, in-depth research should be carried out to support the model. It is recommended that officers should request the following on justifying the CFD simulations in project submission:

(1) The three groups of parameters in the CFD models.
(2) Details of the grid systems. Note that in a large airport terminal, a 10 m grid means that velocity and temperature are the same within 10 m.
(3) The convergence and stability criteria, turbulent parameters.
(4) The boundary conditions on ν, P, T, etc.
(5) The extension of free boundaries.
(6) The prediction and presentation in the following three groups [18] should be evaluated:
   - Velocity-temperature
   - Pressure
   - Turbulent parameters

(7) Justification on the above three groups of CFD predictions by empirical formula and analytical expressions.
(8) Experiments on scale models of the building to compare with CFD predictions.
(9) In-situ field tests to evaluate the system performance.

In summary, officers should request justification on:

- The choice of grid that gives suitably grid-independent results.
- The location of the external boundary conditions that influence the predictions.
- Quality control checks for the CFD predictions to give details of :
  - exchange flow
  - neutral pressure planes
- Experimental verification using data from full-scale burning tests.

6. Immediate Action for Existing Projects

It is no doubt that new projects based on CFD must be assessed with caution, since so many problems have been identified. The suggested evaluation guide is a good starting point.

All existing PBD/FEA projects – which fire hazard assessment was based on CFD – must be reviewed carefully to identify possible problems as CFD has been proven to give inaccurate predictions. It is too harsh to replace engineering hardware such as passive construction protection and active fire engineering system. The only feasible solution is to implement appropriate fire safety management measures to ensure that the assumptions about fire hazard assessment in PBD/FEA reports are followed.

Crowded areas such as railway and subway stations must be monitored vigilantly, particularly those using CFD to predict Available Safe Egress Time (ASET) [3]. FSD should work out appropriate fire safety and rescue strategy, upgrade the equipment and provide more thorough training programmes for firefighters. That means firefighters have to shoulder more tasks than expected. Most importantly, fire inspections must be carried out regularly to ensure that fire safety management is not just a certificate locked in a cupboard.

7. Conclusion

The advantages of CFD have always been hailed, but its problems are rarely discussed in the literature. More justification work should be done to improve further studies on fire protection design and evaluation. CFD is only suitable in FEA after development if it is clearly justified. Therefore, the justification of CFD fire models is necessary before its application. Indiscriminate application would lead to wrong results. Verification and validation studies, such as the works carried out by the US Nuclear Regulatory Commission [13], are important. Full-scale burning tests are necessary in the testing and commissioning stage.

The author has been working on CFD in the past 30 years, and cutting research funding would limit the research of CFD. Trained fire researchers are now working as CFD practitioners due to the extensive application of CFD for FEA projects, and fire science is now deprived of talents. And even engineers without appropriate training in CFD are working as CFD experts. This is another trend that needs our attention, and it will be reported in later articles.

Acknowledgement

The work described in this talk was partially supported by the Construction Industry Institute (Hong Kong) / PolyU Innovation Fund for the project "Assessment of a fire model in simulating combustion for construction projects with fire engineering approach".

References

1. W.K. Chow, "Experience on implementing performance-based design in Hong Kong", Welcome speech, The 9th Asia-Oceania Symposium on Fire Science and Technology, Hefei, China, 17-20 October (2012).
2. W.K. Chow, "The four fire code systems", Department of Building Services Engineering, The Hong Kong Polytechnic University, Hong Kong, February 2013. Available at: http://www.bse.polyu.edu.hk/researchCentre/Fire_Engineering/Hot_Issues.html
3. W.K. Chow, "Letter to the Editor: Comment on 'RSET/ASET, a flawed concept for fire safety assessment' by V. Babrauskas, J.M. Fleming and B.D. Russell, Fire and Materials, Vol. 34, pp. 341-355 (2010)", Fire and Materials, Vol. 37, No. 3, pp. 257-258 (2013).
4. Fire Dynamics Simulator, http://www.nist.gov/el/fire_research/fds_smokeview.cfm
5. Codes of Practice for Minimum Fire Service Installations and Equipment and Inspection Testing and Maintenance of Installation and Equipment, Fire Services Department, Hong Kong Special Administration Region, China (2012).
6. W.K. Chow, "Performance evaluation of atrium smoke exhaust systems with hot smoke tests", SciTopics, February (2009). http://www.scitopics.com/Performance_Evaluation_of_Atrium_Smoke_Exhaust_Systems_with_Hot_Smoke_Tests.html
7. G. Cox, Combustion Fundamentals of Fire, London; San Diego: Academic (1995).
8. G.W. Zou, H.P. Tan, W.K. Chow and Y. Gao , "Effects of varying Smagorinsky constant on simulating post-flashover fires", International Journal of Computational Fluid Dynamics, Vol. 21, No. 2, pp. 107-119 (2007).
9. T.G. Ma, J.G. Quintiere. Numerical simulation of axi-symmetric fire plumes: accuracy and limitations. Fire Safety Journal, Vol. 38. pp. 467-492 (2003).
10. W.K. Chow, CPD lecture "Common mistakes made in using CFD in FEA projects", Organized by Research Centre for Fire Engineering, Department of Building Services Engineering, The Hong Kong Polytechnic University, 21 April (2012).
11. W.K. Chow, "Application of computational fluid dynamics for simulating building fires in subway system: Current situation", In: G. Meschke, J. Eberhardsteiner, T. Schanz, K. Soga and M. Thewes (Editors), Proceedings of the Third International Conference on

Computational Methods in Tunnelling and Subsurface Engineering (EURO:TUN 2013), Ruhr University Bochum, Germany, 17-19 April 2013, pp. 701-712.

12. J. Hietaniemi, S. Hostikka and J. Vaari, “FDS simulation of fire spread comparison of model results with experimental data”, VTT Working Papers 4 (2004).
13. K. Hill, J. Dreisbach, F. Joglar, B. Najafi, K. McGrattan, R. Peacock, and A. Hamins. Verification and Validation of Selected Fire Models for Nuclear Power Plant Applications. NUREG 1824, United States Nuclear Regulatory Commission, Washington, DC, USA (2007).
14. K. Hill, J. Dreisbach, F. Joglar, B. Najafi, K. McGrattan, R. Peacock, and A. Hamins. Verification and Validation of Selected Fire Models for Nuclear Power Plant Applications. NUREG 1824, United States Nuclear Regulatory Commission, Washington, DC. (2007).
15. R.N. Mawhinney, E.R. Galea, N. Hoffmann and M.K. Patel, “A critical comparison of a PHOENICS based fire field model with experimental compartment fire data”, Journal of Fire Protection Engineering, Vol. 6, No. 4, pp. 137-152 (1994).
16. A. Schaelin, J. van der Maas and A. Moser, “Simulation of airflow through large openings in buildings”, ASHRAE Transactions, Vol. 98, No. 2, pp. 319-328 (1992).
17. A. Jowsey, “The process of fire modelling”, International Fire Professional - The Journal of the Institution of Fire Engineers, Issue no. 4, pp. 16-19, April (2013).
18. Y.Q. Chen, “CFD for simulating air distribution in buildings: The state of the art, challenges, and opportunities”, Proceedings of the 11th International Conference on Air Distribution in Rooms (ROOMVENT 2009), 24-27 May 2009, Busan, Korea, pp. 23-31 (2009).

HIFECFD1l

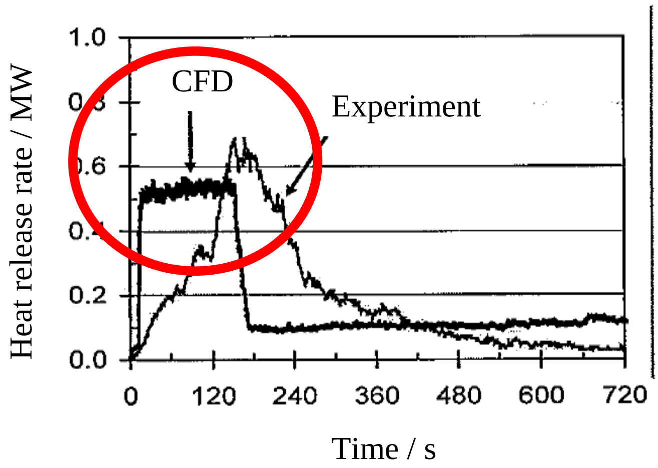

Fig. 1: A bad example of CFD prediction, Hietaniemi et al. (2004)

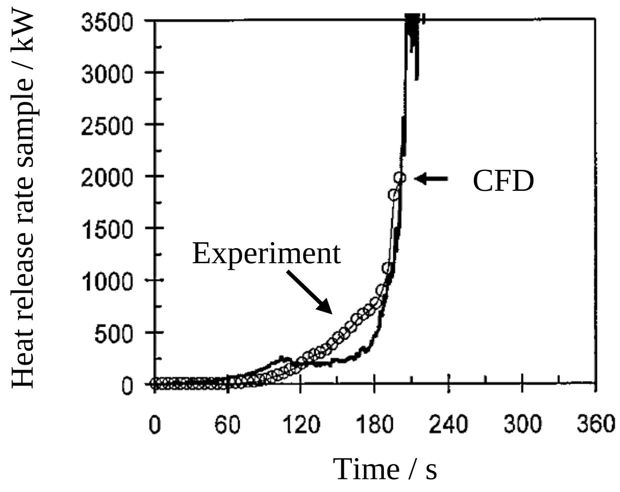

Fig. 2: A good CFD prediction, Hietaniemi et al. (2004)

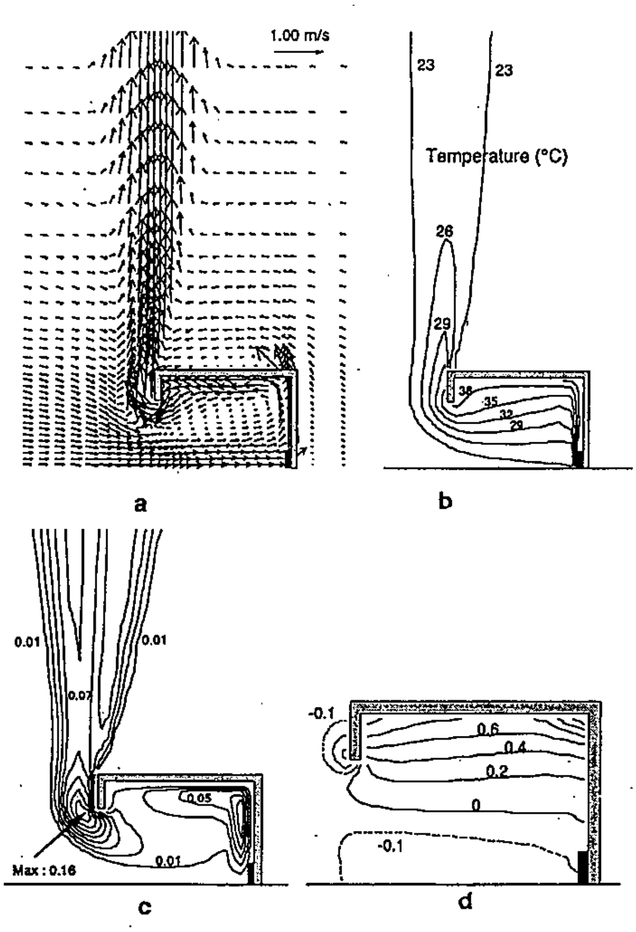

Fig. 3: Extended free boundaries by Schaelin et al. (1992)