

CFD History and Applications

Essam E. Khalil ^C

*Professor of Mechanical Engineering, Faculty of Engineering
Cairo University, EGYPT*

CFD techniques were developed over the years with the hard way of trial and error, refine and many validation and assessment procedures. In the early 1973 the CFD group at Imperial College embarked on an ambitious and attractive program to predict simple shear flows, free and confined jet flows. The work was very intuitive and with modest attempts to predict flow pattern with two and then three dimensional flow configurations. These mainly relate to simple parabolic flow with no recirculation and using stream function –vorticity solution algorithm, hence yielding some non-measurable flow characteristics that made the application rather uncomfortable to compare to real engineering problems. Later that year the newly proposed SIMPLE Semi Implicit solution algorithm partially paved the way with primitive variables velocity U and Pressure P as main parameters . This enables the solution of the Navier Stokes Momentum Equations in a straightforward manner. Grid sized started by 20x20 and up to 10000 orthogonal nodes to converge in 1000 iterations for simple confined symmetrical pipe flow. Necessarily, a model to represent the turbulent characteristics of the flow at high Reynolds numbers was developed by Launder and Spalding in 1974. That was the birth of what is commonly known today as the Standard k - ϵ turbulence model; also known as two equation turbulence model.

One should bear in mind that this model was originated based on constants derived from simple shear flow measurements. However, the application of this 1st generation eddy viscosity model is now extrapolated to swirling, reacting complex geometries with strong turbulence chemistry interactions. The success story of that model encouraged everyone around the world to test validates that model. Among others, Gosman, Khalil and Whitelaw in 1977 had published an article with systematic validation of k - ϵ model in a variety of applications with and without swirl. Khalil, Spalding and Whitelaw in 1975 published their validation of that turbulence mode in a turbulent reacting furnace configuration with and without swirl.

In Early 1986 Prof. D.B. Spalding gave a general lecture presentation on Computational fluid dynamics in engineering and education at Rensselaer Polytechnic Institute Sadowsky Series, New York,USA. The

^C Email: khalilel@asme.org

© 2009-2012 All rights reserved. ISSR Journals

PII: S2180-1363(12)4243X

basics of computational fluid dynamics (CFD) are recalled, and the place of CFD in engineering and education is described. Software and hardware requirements were elegantly discussed. By that time one can assume that the first stage of development was carried out with remarkable results that built bridges of confidence between researchers and the used on the application side (industry and Power generation sectors). Many Industrial applications and modelling of furnaces and combustion chambers were successfully carried out utilizing various forms of the Imperial College and Spalding's basic concept. The main problems facing the CFD at that time were:

- Computational limitations on mainframe
- Difficulty to handle irregular boundary and wall conditions with simple orthogonal grid.
- Slow convergence and numerical diffusion.
- Difficulty to carry out three dimensional complex geometries.
- Time dependent calculations were still in the cradle.

With the birth of personal computers with escalating memory capacities, business became easier, but communications between researchers at various institutes became less as most were confined to the PC in their own offices and then their own homes when laptops appear. The software providers had excelled their efforts producing, more enduring and user friendly programs with detailed manuals of operations.

Recent development over the past two decades has shown wide variety of applications in combustor designs, aerodynamic simulation of aerospace flights and shuttles as well as other aerospace applications. CFD were extensively used for power plants simulations such as thermal patterns of boiler furnaces, turbine blading performance as well as heat exchangers designs.

With the need to create a more energy sustainable environment, the attention was focused to built environment with aim of ultimately reducing carbon metrics and energy use in Buildings. A wide variety of packaged software programs were devised and used by experts such as Energy Plus and Designbuilder. The literature is full now with many publications that cover all scopes of energy utilization in built environment.

In the past decades, Computational Fluid Dynamics (CFD) has been studied intensively as a tool for evaluating the indoor environment of buildings and its interaction with the building envelope, as well as for analysing the outdoor environment around buildings. While the use of CFD in engineering practice is becoming quite well established for the wide variety of indoor applications, this is considerably less pronounced for outdoor applications. In complex case studies, wind environmental problems such as pedestrian wind nuisance and air pollutant dispersion are still typically investigated in boundary layer wind tunnels, while wind-driven rain exposure and convective heat and mass transfer

coefficients at building surfaces are generally estimated from simplified empirical or semi-empirical formulae. An important disadvantage of wind tunnel measurements however is that usually only point measurements are obtained. Techniques such as Particle Image Velocimetry (PIV) and Laser-Induced Fluorescence (LIF) in principle allow planar or even full 3D data to be obtained, but the cost is considerably higher and application for complicated geometries is hampered by laser-light shielding by the obstructions constituting the urban model.

Another disadvantage is the required adherence to similarity criteria in reduced-scale testing. This can be a problem for, e.g., multiphase flow problems and flows in which density differences are an important driving force. Examples are wind-driven rain and pollutant dispersion studies. Empirical and semi empirical formulae generally only provide a first, crude indication of the relevant parameters, often in averaged form (e.g., surface-averaged) or at a few discrete positions. The information they provide is often too simplified compared to the well-established building performance simulation tools in which this information is used. Examples are wind-driven rain intensities and convective heat and mass transfer coefficients for building envelope Heat-Air-Mass (HAM) transfer tools and Building Energy Simulation (BES) software. Numerical modelling with CFD could be a powerful alternative because it can avoid some of these limitations. It can provide detailed information on the relevant flow variables in the whole calculation domain (“whole-flow field data”), under well controlled conditions and without similarity constraints. However, the accuracy of CFD is an important matter of concern. Care is required in the geometrical implementation of the model, in grid generation and in selecting proper solution strategies. In addition, numerical and physical modelling errors need to be assessed by detailed verification and validation studies. The open literature included a brief, non-exhaustive overview of the status of the application of CFD in building performance simulation for the outdoor environment. It focuses on four topics: (1) pedestrian wind environment around buildings; (2) wind-driven rain on building facades; (3) convective heat and mass transfer coefficients at building surfaces; and (4) air pollutant dispersion around buildings. For each topic, some specific difficulties, advantages and disadvantages of CFD were also discussed.

References

- Launder, B.E. and Spalding, D.B., 1974, The Numerical Computation of Turbulent Flows, *Computer Methods App. Mech.*, pp. 269-275.
- Gosman, A.D., Khalil, E.E. and Whitelaw, J.H., 1977 “The Calculation Of Two Dimensional Turbulent Recirculating Flows”, *Proc. 1st*

Symposium On Turbulent Shear Flows, 1, 13.35, See Also Turbulent Shear Flows, pp .237-255, 1979.

Khalil, E.E., Spalding, D.B. and J.H.Whitelaw, J.H., 1975, “The Calculation of Two Dimensional Furnaces”, *Int. J. Heat & Mass Transfer*, 18, pp.775-791.

Khalil, E.E., 2012, “Holistic Approach to Energy Efficient Green Buildings from Construction Material to Services”, Invited paper, TTMD 10th International Symposium, Istanbul, May 2012.