Sazhin, Irrigat Drainage Sys Eng 2012, 1:4
DOI: 10.4172/2168-9768.1000e110

Irrigation & Drainage Systems Engineering

ISSN: 2168-9768

Editorial

Open Access

Computational Fluid Dynamics: A Tool for Engineering Research?

Sergei S. Sazhin*

Sir Harry Ricardo Laboratories, Centre for Automotive Engineering, School of Computing, Engineering and Mathematics, Faculty of Science and Engineering, University of Brighton, Brighton, UK

Editorial

The importance of Computational Fluid Dynamics (CFD) codes in modelling of the processes in various engineering systems, including those related to irrigation and drainage is well recognized [1]. The range of problems to which these codes can be applied and the reliability of their predictions, however, are still far from clear to the engineering community, although the developments of these codes led to their application to new fields, including modelling of magneto hydrodynamic processes and the processes in rarefied gases [2,3]. At first sight the main problem here lies in the limitations of the available computer power, and application of direct numerical simulations could make the predictions of these codes almost 100% reliable. This could be potentially achieved only in the case when we are able to determine very accurately all initial and boundary conditions. This is obviously not possible in most realistic engineering systems. Hence, any predictions of CFD codes are always expected to be approximate. In what follows the nature of these approximations and possible pitfalls in the interpretation of CFD results are discussed.

A widely used simplification in CFD analysis is based on the reduction of the dimensions. As an example, we can consider flow around an infinitely long cylinder or square prism, perpendicular to the direction of the flow. At first sight this problem could be considered based on the 2-dimensional (2D) approximation [4]. However, at Re>200, vortices developing behind the cylinder become unstable to spanwise bending [5] which cannot be modelled based on the 2D approximation.

Another widely used simplification is based on the assumption that the effects of turbulence in the flow can be described by the Reynolds Average Navier-Stokes (RANS) equation. This approach, however, cannot explain the origin of noise (flow induced vibrations) in the flow [5].

A whole range of new problems emerges when we attempt to apply CFD codes to modelling multiphase flows. In the case of single-phase flows the reduction of the cell sizes is expected to always improve the accuracy of calculations. This is not always the case when multiphase flows are modelled and special techniques need to be applied to eliminate the grid dependence of the results of this modelling [6].

Many practically important processes, including stability analysis of the flows, cannot be described with the help of any available CFD code [7].

To summarize the above brief analysis, we can conclude that CFD codes cannot be used as exclusive tools for engineering research. They can, however, be useful tools in this research if used together with other tools of flow analysis. We need to have a clear idea about the general properties of the flow to be analysed before a CFD code can be applied to the analysis of the details of this flow. The link between CFD codes and other tools used for the analysis of the flows is still an open question.

In some cases CFD analysis can be complemented by asymptotic analysis [8]. Also, CFD results can be complemented by more advanced models of individual processes [9]. Recently, a new direction in CFD code developments, based on direct implementation of new analytical solutions into these codes, was developed [10,11].

References

1. Lorezini G, Conti A, De Wrachien DG (2012) Computational Fluid Dynamics (CFD) picture of water droplet evaporation in air. Irrigat Drainage Sys Eng 1: 1-12.
2. Ishii T, Sazhin SS, Makhlouf M (1996) Numerical prediction of magnetohydrodynamic (MHD) flow in continuous casting process. Ironmaking and Steelmaking 23: 267-272.
3. Sazhin SS, Serikov VV (1997) Rarefied gas flows: hydrodynamic versus Monte Carlo modelling. Planet Space Sci 45: 361-368.
4. Pavlov AN, Sazhin SS, Fedorenko RP, Heikal MR (2000) A conservative finite difference method and its application for the analysis of a transient flow around a square prism. Int J Numer Meth Heat Fluid Flow 10: 6-46.
5. Panton RL (1997) Incompressible Flow. John Wiley & Sons, Chichester, UK.
6. Tonini S, Gavaises M, Theodorakakos A (2008) Modelling of high-pressure dense diesel sprays with adaptive local grid refinement. Int J Heat Fluid Fl 29: 427-448.
7. Boronin SA, Healey JJ, Sazhin SS (2013) Nonmodal stability of round viscous jets. J Fluid Mech (in press).
8. Sazhin SS, Shchepakina E, Sobolev V (2010) Order reduction of a non-Lipschitzian model of monodisperse spray ignition. Math Comput Model 52: 529-537.
9. Abramzon B, Sazhin S (2005) Droplet vaporization model in the presence of thermal radiation. Int J Heat Mass Tran 48: 1649-1882.
10. Sazhin SS, Krutitskii PA, Gusev IG, Heikal MR (2010) Transient heating of an evaporating droplet. Int J Heat Mass Tran 53: 2826-2836.
11. Mitchell SL, Vynnycky M, Gusev IG, Sazhin SS (2011) An accurate numerical solution for the transient heating of an evaporating spherical droplet. Appl Math Comput 217: 9219-9233.

***Corresponding author:** Sergei S. Sazhin, Sir Harry Ricardo Laboratories, Centre for Automotive Engineering, School of Computing, Engineering and Mathematics, Faculty of Science and Engineering, University of Brighton, Brighton, BN2 4GJ, UK, E-mail: S.Sazhin@brighton.ac.uk

Received October 30, 2012; **Accepted** October 30, 2012; **Published** November 03, 2012

Citation: Sazhin SS (2012) Computational Fluid Dynamics: A Tool for Engineering Research? Irrigat Drainage Sys Eng 1:e110. doi:10.4172/2168-9768.1000e110