Open forum

EVALUATE SHOCK CAPTURING CAPABILITY WITH THE NUMERICAL METHODS IN OpenFOAM

by

Reza KHODADADI AZADBONI [a*]***, Mohammad Rahim MALEKBALA*** [b]***, and Fatemeh KHODADADI AZADBONI*** [a]

[a] Young Researchers Club, Sari Branch, Islamic Azad University, Sari, Iran
[b] Young Researchers Club, Science and Research Branch, Islamic Azad University, Tehran, Iran

Original scientific paper
DOI: 10.2298/TSCI130425048K

Simulations for both multiphase flows and supersonic single phased flows are well known, however the combination is a less investigated area of research, as the two basic approaches of computational fluid dynamics, the pressure and the density based approach, each describe one of the phases in a better way than the other one. In this paper, we systematically investigate the solver quality of the open source computational fluid dynamics code OpenFOAM in handling transonic flow phenomena that typically occur inside the breaking chamber of high voltage circuit breakers, during contact separation. The solver quality is then compared with that of chosen commercial computational fluid dynamics tools. The main advantage of OpenFOAM is that, contrary to most of the commercial simulation tools, it is license fee free and allows access to the source code. This means that complicated multi physics phenomena inside the arcing chamber can be directly modeled into the code by users, which opens an opportunity to remove limitations of commercial computational fluid dynamics tools. Particularly, the shock capturing capability of OpenFOAM will be evaluated for the transonic internal flow which typically occurs in high voltage circuit breakers. Overall, OpenFOAM shows acceptable shock capturing capabilities in the performed verification and validation studies, with the solver quality comparable to some of the tested commercial computational fluid dynamics tools. There is still room for further solver quality improvements in OpenFOAM by implementing better shock capturing schemes such as a density-based flux-difference-splitting scheme or by writing better physical modeling of the shock/boundary layer interaction into the open architecture of OpenFOAM.

Key words: *OpenFOAM, computational fluid dynamics, shock capturing, density based, pressure based*

Introduction

In the design and development of high voltage circuit breakers, it is important to understand the dynamics of the transonic gas flow that is used to extinguish the arc between the contacts. It is nowadays routine engineering practice in high voltage circuit breaker development to use computational fluid dynamics (CFD) based simulation technologies. The application of the reliable CFD technology can save a number of type tests and significantly accelerate the prototyping, which his directly connected to development cost reduction.

* Corresponding author; e-mail: reza.khodadadi@modares.ac.ir

There are several features that the ideal CFD simulation tool must possess for high voltage circuit breaker development. Since the huge pressure build up in the heating volume induced by the electric arc power generates transonic flow in the diffuser area, the CFD tool must be able to capture the key physical features of the transonic nozzle flow. These are characterized by the qualities of capturing the exact shock location, shock induced flow separation from the diffuser wall and the extension of the re-circulation zone.

The ultimate goal of the adopted CFD tool for breaker development would be to include multi-physics simulation features such as electro-magneto-hydrodynamics. This will then enable the simulation of the whole circuit breaking process. From this viewpoint, the CFD tool should either offer all these multi-physical features or at least be extensible to that effect.

In this paper, we evaluate the CFD software OpenFOAM [1], which should provide with unlimited extensibility as an open source code. In particular, the quality features of shock capturing and shock-induced flow separation will be evaluated. For this purpose, we carry out two verification studies where the exact solutions are known, and two validation studies for two transonic internal nozzle flow experiments. Also, the solution quality of the OpenFOAM simulations is compared with that of selected commercial CFD tools, one based on the semi-implicit pressure-correction based SIMPLEC method and the other based on a density-based flux-difference-splitting scheme, selectively with explicit and implicit time marching [2].

In this paper, all the simulations are performed with second order space accuracy. The transonic cold air flows are simulated by numerically solving either Euler or Navier-Stokes equations, with laminar setting. The purpose of the verification study is to evaluate the shock capturing capability of the numerical method and to check whether it solves the Euler equation accurately. In this section, we choose two standard verification cases from literature [3], where the numerical solution can easily be compared with the exact solution of the Euler equations.

The shock tube is a quasi-1-D problem with discontinuous initial conditions. A high-pressure area is initially separated from a low pressure area by a diaphragm [4]. Once the diaphragm is removed, the shock and the contact discontinuity begin to travel into the initially low pressure region, whereas the rarefaction wave travels into the initially high pressure region, see fig. 1 and [5-8].

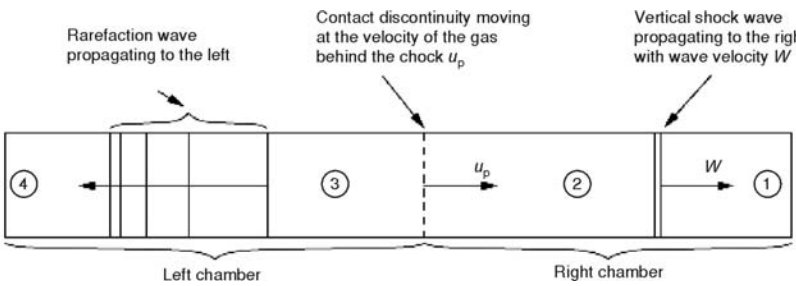

Figure 1. Shock tube after the diaphragm is broken [3]

Exact solutions

The derivation of an exact solution to the shock tube problem can be found in any textbook on compressible flows. Here, we follow Anderson, omitting the parts about reflected shock and expansion waves as well as some theory considered too general [3]. Here we just mention the final equation of exact solution, that done by Anderson:

$$\frac{p_2}{p_1}\left[1-\frac{(\gamma_4-1)\left(\dfrac{a_1}{a_4}\right)\left(\dfrac{p_2}{p_1-1}\right)}{\sqrt{2\gamma_1\left[2\gamma_1+(\gamma_1+1)\left(\dfrac{p_2}{p_1-1}\right)\right]}}\right]^{-2\gamma_4/(\gamma_4-1)}-\frac{p_4}{p_1}=0 \tag{1}$$

Equation (1) is solved using the Matlab function Fzero, which employs a combination of bisection, secant, and inverse quadratic interpolation methods to find a zero γ – specific heat ratio, a_1-a_4 – constants). As an initial guess, $(p_4/p_1)/2$ is taken. From this, the pressure in region 2, p_2, is obtained. With the pressure ratio p_2/p_1 at hand, all other shock properties can be calculated. Since pressure and velocity are constant across the contact surface between regions 2 and 3, we know p_3, u_2, and u_3; using these, the other quantities in region 3 can be determined [9].

Numerical scheme

For the CFD simulations, for comparison and the investigation of quality of the multiphase solver for highly compressible flows, the solver is used as a single phased solver by setting the same properties for both of the fluids. We consider a shock tube of infinite length where the diaphragm is located at $x = 1.524\cdot10^{-1}$ m. The initial condition is described by $p_1 = 6.897\cdot10^4$ Pa, $T_1 = 231.11$ K for the right chamber and $p_4 = 6.897\cdot10^4$ Pa, $T_4 = 288.89$ K for the left chamber as it described in tab. 1.

Table 1. Initial values for the shock tube problem

Compartment	Pressure [Pa]	Temperature [K]
Left (driver)	$p_4 = 6.897\cdot10^4$	$T_4 = 288.89$
Right (driven)	$p_1 = 6.897\cdot10^3$	$T_1 = 231.11$

In tab. 2, the Numerical Schemes methods, uses in OpenFoam, are described.

Table 2. The definitions of the used solvers

Solver	Definition
rhoCentralFoam	Density-based compressible flow solver based on central-upwind schemes of Kurganov and Tadmor
rhopSonicFoam	Pressure-density-based compressible flow solver
rhoSonicFoam	Density-based compressible flow solver
sonicFoam	Transient solver for trans-sonic/supersonic, laminar or turbulent flow of a compressible gas (pressure based)
compressibleInterFoam	Solver for 2 compressible, isothermal immiscible fluids using a VOF (volume of fluid) phase-fraction based interface capturing approach (pressure based)

Results and discussions

Results show the pressure wave structure obtained from OpenFOAM and other benchmark CFD solvers, by running transient simulations for $t = 225$ μs. The numerical solutions are also compared with the exact solution. The used computational setting for OpenFOAM is the pressure-correction based PISO algorithm. The compared benchmark commercial solvers

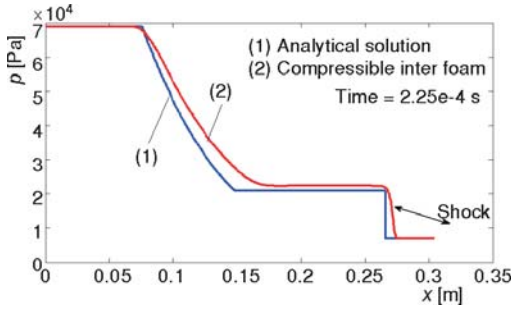

Figure 2. Pressure distribution in the shock tube at t = 225 μs; for 500 cell

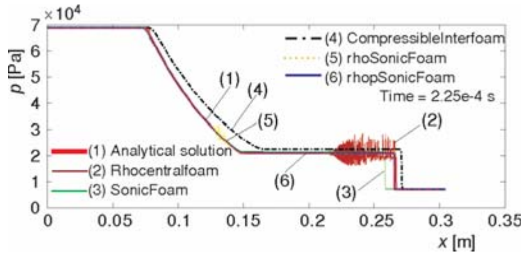

Figure 3. Pressure distribution in the shock tube at t = 225 μs; for 4000 cell

are, respectively, based on the pressure-correction based SIMPLEC method and the density-based Roe scheme with non-iterative implicit time marching.

First we have done the numerical solution with mesh that contain 500 cell, and with attention to fig. 2, the sharpness of shock in the figure of the exact solution and numerical was not as same as each other, because of this we should finer the mesh size, and finally we choose 4000 cell for the numerical solution, that shown in fig. 3.

The first observation is that rhopSonicFoam performs best of all solvers: it does not exhibit any overshoots and only little numerical dissipation. SonicFoam has no overshoots either, but clearly the most numerical dissipation of the three laminar solvers. RhoSonicFoam lies in between in that respect but seems to be a victim of numerical dispersion, as the wiggles in the neighborhood of large gradients indicate.

In compressibleInterFoam, that due to the simplified physics in the multiphase solver. There is no energy equation solved and thus compressibility is constant for every time step at every location. At high pressures temperature dependency of compressibility $\psi(\rho = \psi(T)\cdot p)$ and also other thermo physical properties has to be considered. Without it there is no calculation of the density jump along the contact surface between the driving and driven fluid.

In rhoCentralFoam the temperature dependent compressibility $\psi(T)$ is implemented. This effect is not used in the multiphase solver, thus the density here is overestimated in the driven fluid and the pressure is higher, too. However along the contact surface the density is supposed to jump.

Therefore for solving such a shock tube problem for two phase flow, with these result it would be apparent that, we should combine a solver such rhopSonicFoam or SonicFoam that solve compressible single phase fluid, with compressibleInterFoam (a solver with two phase interface tracking method). Thus a new OpenFoam solver (that in this research called CompressibleVOFFoam) with the basis of compressibleInterFoam solver with adding energy equation had created. In the following, Sod's shock tube problem will be present for this new compressible two phase solver (CompressibleVOFFoam).

Sod problem

The shock tube problem which was used by Sod [10] to test a number of methods for solving the equations of compressible flow, has become a standard test problem. The initial conditions for this problem consist of two semi-infinite states separated by a diaphragm at time $t = 0$ [10]. The left and right states are set to the following conditions (tab. 3).

Table 3. Initial condition of Sod's problem

Compartment	$X > 0.5$ Left (driver)	$X < 0.5$ Right (driven)
Pressure	$p_L = 1$	$p_R = 0.1$
Density	$\rho_L = 1$	$\rho_R = 0.125$
Velocity	$U_L = 0$	$U_R = 0$

With $\psi = 1.4$ The result of a numerical simulation of Sod's problem at time $t = 225$ μ second obtained from this code is compared with exact solution in figS. 4 to 7. To test the grid dependency, the calculations were repeated with 50, 100, and 200 grid point. Figure 4 shows that the calculation with 200 grid points is converged to the exact solution. This calculation has been performed in a constant mesh domain. The capability of the present code to capture shock and other discontinuities was quite good.

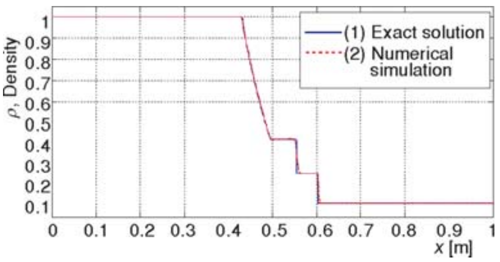

Figure 4. Comparison of density distribution between the exact solution and numerical simulation for Sod's problem ($t = 2.25 \cdot 10^{-4}$ s, $\Delta x = 0.01$)

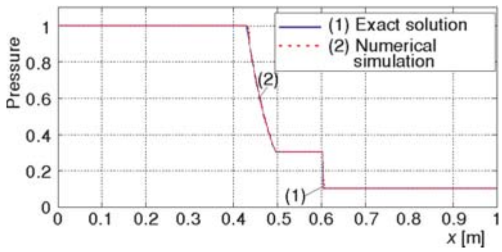

Figure 5. Comparison of pressure distribution between the exact solution and numerical simulation for Sod's problem ($t = 2.25 \cdot 10^{-4}$ s, $\Delta x = 0.01$)

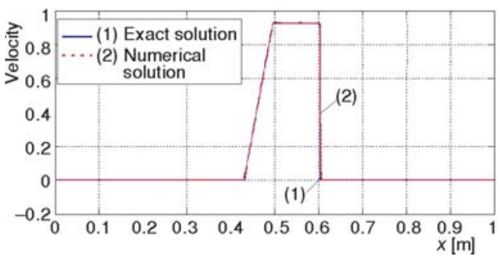

Figure 6. Comparison of velocity distribution between the exact solution and numerical simulation for Sod's problem ($t = 2.25 \cdot 10^{-4}$ s, $\Delta x = 0.01$)

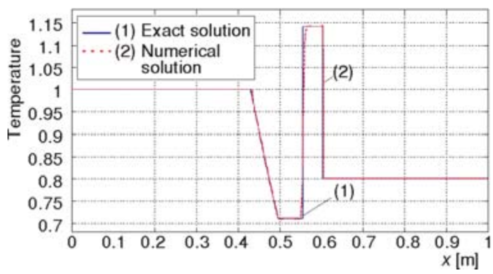

Figure 7. Comparison of temperature distribution between the exact solution and numerical simulation for Sod's problem ($t = 2.25 \cdot 10^{-4}$ s, $\Delta x = 0.01$)

Conclusions

In this paper, we have investigated the solver quality of OpenFOAM in handling transonic internal flow phenomena which typically occur inside high voltage circuit breaker diffusers. OpenFOAM is based on a pressure-correction based PISO algorithm and has shown acceptable shock capturing capabilities in the presented verification and validation studies. The shock capturing capability of OpenFOAM has been compared with that of chosen commercial

CFD solvers. While the CFD solver based on a pressure-correction based SIMPLEC method has shown comparable shock capturing capability to OpenFOAM, the density-based explicit solver has shown better shock capturing quality. This indicates that the implementation of the density-based flux-difference splitting and/or the flux-vector-splitting schemes into the open architecture of OpenFOAM could improve its shock capturing capability further. It can be said, that OpenFOAM is capable of depicting a multiphase flow, it is also is capable of handling a supersonic flow without bigger problems. The combination of these two is however assured just for slightly or semi-compressible fluids. Attempts in the future have to be done in order to integrate the energy equation into the multiphase solver of OpenFOAM for better quality in the investigation of supersonic multiphase flows.

References

[1] ***, OpenCFD, OpenFOAM: The Open Source CFD Toolbox. User Guide Version 1.4, OpenCFD Limited, Reading UK, 2007
[2] ***, OpenCFD Ltd. http://www.opencfd.co.uk/openfoam/standardSolvers.html#standardSolversb OpenCFD Ltd., 2004-2009
[3] Anderson, Jr., John D, *Modern Compressible Flow: With Historical Perspective*, 3rd ed. McGraw-Hill, New York, USA, 2003
[4] Bogar, T. J., *et al*., Characteristic Frequencies of Transonic Diffuser Flow Oscillations, *AIAA Journal, 21* (1983), 9, pp. 1232-1240
[5] Hsieh, T., *et al.*, Numerical Investigation of Unsteady Inlet Flow Fields, *AIAA Journal, 25* (1987), 1, pp. 75-81
[6] Georgiadis, N. J., *et al*., Evaluation of Turbulence Models in the Parc Code for Transonic Diffuser Flows, Technical Memorandum 106391, NASA Lewis Research Center, Ohio, USA, 1994
[7] Salmon, J. T., *et al*., Laser Doppler Velocimeter Measurements in Unsteady, Separated, Transonic Diffuser Flows, *AIAA Journal, 21* (1983), 12, pp. 1690-1697
[8] Mantilla, J. D., *et al*., Measurements and Simulations of Cold Gas Flows in High Voltage Gas Circuit Breakers Geometries, *Proceedings,* IEEE International Symposium on Electrical Insulation (ISEI 2008), Vancouver, Canada, 2008
[9] Wuthrich, B., Lee, Y., Verification and Validation Studies of OpenFOAM for Transonic Compressible Flow Simulations Inside High Voltage Circuit Breaker Diffusers, CH-5405 Baden-Dattwil, Switzerland, IEEE, 2008
[10] Sod, G. A., A Numerical Study of Converging Cylindrical Shock, *J. Fluid Mech., 83* (1977), 4, pp. 785-794

Paper submitted: August 2, 2011
Paper revised: April 25, 2013
Paper accepted: April 25, 2013