Review and comparison of two OpenFOAM® solvers: rhoCentralFoam and sonicFoam

Maria Laura Canteros¹*, and Jiří Polanský²,¹

¹CTU in Prague, Faculty of Mechanical Engineering, Department of Fluid dynamics and Thermodynamics, Technická 4, 160 00 Prague, Czech Republic
²MECAS ESI s.r.o., Brojova 2113, 326 00 Plzeň 2, Czech Republic

Abstract. This article deals with the comparison and review of two solvers from the open-source CFD tool, OpenFOAM®, in the field of compressible fluids: the explicit segregated density-based solver (rhoCentralFoam), and the PIMPLE algorithm solver (sonicFoam). In spite of the numerous studies showing both solvers, there are not yet been presented complete flowcharts. This work will expose the creation of two flowcharts based on bibliographic review. For the purpose of the demonstration of the application of the two solvers, the Mach number, and the mass flow rate in time in the current case analysis are numerically estimated in both cases in an ISO 9300 Venturi nozzle case analysis.

1 Introduction

Incompressible fluids are an area of interest where Mach number, dimensionless number defined by the ratio between the velocity and the local speed of sound, is smaller than 0.3. Even if the assumption of a fluid as incompressible radically reduces the computation time and resources, almost all fluids present some element of compressibility at a certain point, particularly for jet engines, high-speed fluids, and rocket motors, among many other applications.

A classic and clear example of subsonic-transonic-subsonic behavior is an ISO 9300 [1] Venturi nozzle, where the incompressible assumptions do not permit an accurate analysis of fluid flowing and is thus, necessary to contemplate a compressible behavior.

Today’s computation requirements are very advanced, and this permits CFD to model complex geometries and to solve the Navier-Stokes equations in a numerical way, avoiding simplifications in order to represent the phenomena in a more realistic way.

Either a new method is conceived or a modification of a current one is implemented; it is necessary to perform a scrutinious analysis, in order to evaluate their accuracy.

In spite of the numerous studies [2–6] that explain the structure of rhoCentralFoam and sonicFoam and compares their performances, there is not yet any flowchart that breaks down and exposes the assembly of any of these solvers. For this purpose, this paper proposes a flowchart of rhoCentralFoam and another flowchart for sonicFoam.

At the same time, as a matter of exemplification, an ISO 9300 cylindrical Venturi nozzle is numerically simulated with both solvers separately, and the results are presented, showing Mach number along the symmetrical section and the flow rate in and out vs time.

2 OpenFOAM®

OpenFOAM® [7] is a free open-source CFD tool that permits the modification of its source code in C++ for different purposes. OpenFOAM® uses a finite volume cell-centered discretization of the domain. The solutions of the governing equations of compressible fluids can be attained either with pressure-based or density-based method.

2.1 rhoCentralFoam

rhoCentralFoam is an explicit segregated density solver for high-speed compressible flow, based on a central-upwind scheme in an alternative approach to Riemann solvers, which is a combination of central-difference and upwind schemes [8,9], where the continuity (1), momentum (2) and energy (3) equations are discretized separately to obtain three linear systems.

\[
\frac{\partial (\rho U)}{\partial t} + \nabla \cdot \left( \rho U U \right) = 0, \\
\frac{\partial (\rho U)}{\partial t} + \nabla \cdot \left( \rho U U \right) + \nabla p + \nabla \cdot \sigma = 0, \\
\frac{\partial (\rho E)}{\partial t} + \nabla \cdot \left( \rho E U \right) + \nabla \cdot \left( \sigma \cdot U \right) + \nabla \cdot j = 0
\]

for density, \( \rho \), velocity, \( U \), pressure, \( p \). The total energy is obtained by \( E = e + \frac{1}{2} |U|^2 \) and the term \( e \) is the specific internal energy or enthalpy, \( \sigma \) is the viscous stress tensor, defined as positive in compression,

\[
\sigma = -\mu \left[ \nabla U + \left( \nabla U \right)^T - \frac{2}{3} (\nabla \cdot U) I \right]
\]

* Corresponding author: maria.laura.canteros@cvut.cz

© The Authors, published by EDP Sciences. This is an open access article distributed under the terms of the Creative Commons Attribution License 4.0 (https://creativecommons.org/licenses/by/4.0/).
with dynamic viscosity $\mu$, $\textbf{I}$ the unit tensor and $\textbf{j}$ is the diffusive flux of heat with thermal conductivity $k$ and temperature $T$,

$$\textbf{j} = -k \nabla T$$

(5)

2.2 sonicFoam

sonicFoam uses the pressure and velocity as dependent variables and is a semi-implicit segregated pressure-based solver, by the means of the PIMPLE algorithm. This algorithm is a combination between the PISO algorithm [10] (Pressure Implicit with Splitting of Operator) and the SIMPLE algorithm.

Pressure equation may be derived from the differential form of the momentum and continuity equations. One of the possible inconsistencies can be the discretization of the corresponding terms in the parent equations with the discretization of all terms of pressure equation. In consequence, the pressure thus obtained may not always yield a velocity field which satisfy simultaneously the momentum and continuity equations.

In the view of the fact that the requisite of the solution of the pressure equation is the satisfaction of the continuity equation it is not possible to accept this incompatibility.

By splitting of the solution process into a series of steps where the operations on pressure are decoupled from those on velocity, the pressure equation may be derived from the discrete forms of the momentum and continuity equations.

One of the forms of representing the spatial and temporal derivatives in the momentum and continuity equations (parent equations) of the finite difference transport equations is using the Euler implicit difference scheme.

In consequence, the difference form for each point of the mesh expressed for compressible flows in the governing equations (1), (2) and (3) are:

$$\frac{1}{\Delta t} (\rho u_i)_{n+1} - (\rho u_i)_{n} = 0$$

(6)

$$\frac{1}{\Delta t} [(\rho E)_{n+1} - (\rho E)^n] + (\rho E u_i)_{n+1} = -(\rho u_i)_{n+1} - (\tau_{ij} u_j)_{n+1}$$

(7)

$$\frac{1}{\Delta t} (\rho u_i)_{n+1} - (\rho u_i)_{n} = -S_{ij,i} - (p)_{n+1}$$

(8)

where $n$ and $n + 1$ denote successive time levels and $S_{ij,i}$ stands for the finite difference representation of the convective and diffusive fluxes of momentum. The operator $(\_)_i$ is the finite difference equivalent to $\partial / \partial x_i$.

3 Solvers’ diagrams

In Fig. 1. Diagram flowchart of rhoCentralFoam a detailed structure of rhoCentralFoam is confectioned and thus, presented, in order to break down its functioning. The relevant steps of the program are presented, and some variables are further deployed.

For the case of sonicFoam solver, the diagram is presented in Fig. 2. Since the scheme englobes SIMPLE algorithm as much as PISO algorithm, the construction of this diagram was presented in a much simpler manner.

The calling of functions as rhoEqn(), pimple.loop(), UEqn, EEqn, pEqn and the turbulence one, are named but not described because it falls out of the scope of the present work, and they have already been studied and broken down in other works.

4 Nozzle simulation parameters and characteristics

Numeric simulation of critical flow through a nozzle is one of the standard compressible flow testing cases. In this study, the mass flow coefficient estimation as main target and flow velocity field estimation is a secondary parameter of interest.

The present model is considered as axisymmetric geometry of standard D=2mm cylindrical critical nozzle (ISO 9300). Atmospheric stagnation pressure and 300K stagnation temperature was applied as inlet boundary conditions. SST k-omega turbulent model for low Reynolds number parameters is used. Atmospheric pressure on ideal gas (air properties) cased the low Reynolds number parameters on critical point of nozzle. To complete a flow nozzle characteristic required investigation on pressure range between $10^4$ and $10^7$ Pa.

In the present case, calculations start with sonicFoam, applied ramp outlet pressure from 90% to 70% of stagnation pressure, delta time about $1e^{-8}$. After developing of subsonic flow field, rhoCentralFoam was applied with delta time about $1e^{-7}$. Initial relaxation factor 0.3 for energy equation was later increased to 0.9.

5 Results and discussion of the simulations

Calculation with density-based solver (rhoCentralFoam) seems to be more sensitive to initial conditions than the pressure based one (sonicFoam). Fig.3 show a comparison of Mach number distribution for specific time. The slight differences of shock waves structure are not relevant due to the fact, the snapshots are captured for different moment of calculation. On the other hand, it is interesting to note that shock waves are captured as sharper for rhoCentralFoam than sonicFoam calculation.
Fig. 3. Mach number iso-surface, cylindrical critical nozzle D=2mm, pressure ration 0.7.

Fig. 4. Inlet and Outlet mass flow oscillation, using rhoCentralFoam.

Fig. 5. Inlet and Outlet mass flow oscillation, using sonicFoam.

6 Conclusions

This paper presents the flowchart diagrams of the two OpenFOAM solvers for compressible flows: rhoCentralFoam and sonicFoam. The diagrams are described in detailed way in order to help and clarify the lecture of the code of both solvers that, until now where never proposed.

At the same time, this work exposes results of simulations of these diagrams in an ISO 9300 critical cylindrical nozzle of 2 mm diameter. The Mach number along the symmetrical axial, as well as the in and out flow rate vs. time. The numerical simulations are illustrative, for the interested to know, that it is possible to estimate such parameters.

RhoCentralFoam is seen to be more sensitive at the initialization of the calculation, requires a low relaxation factor of energy and a suitable initial velocity field. However, it is just as comfortable to work with as sonicFoam, which is not as sensitive to initialization at first. RhoCentralFoam shows sharper shock waves, but otherwise the velocity field is very similar to sonicFoam results. The calculation of the flow coefficient is not quite the same, but within the expected range of CFD.

This work was supported by the Grant Agency of the Czech Technical University in Prague, grant No. SGS22/099/OHK2/2T/12.

References

7. https://www.openfoam.com/, (n.d.)