5th World Conference on
Applied Sciences, Engineering & Technology
02-04 June 2016, HCMUT, Vietnam

Numerical Simulation of Supersonic Flow and Shockwave Using OpenFOAM

HOAI NGUYEN, MINH MAN PHAM, LE CHAU THANH NGUYEN
Department of Mechanical Engineering, College of Technology, The University of Danang, Da Nang, Viet Nam
Email: hoainguyen.tme@gmail.com

Abstract: Computational fluid dynamics (CFD) is one of the common applications of fluid mechanics, numerical methods and algorithms are used to solve problems related to fluid flow. Nowadays, computers are used to simulate the interaction of liquid and airflow in a region defined by boundary conditions, however, in many cases, the algorithm stops at only approximate. Along with the on-going development of research work, many open source software was created to improve the accuracy and speed in situations simulating complex flows. In this paper, supersonic flow and shockwave are simulated in a flat domain size determined by using OpenFOAM. Since then, the theoretical basis of the problem is built, while some parameters, such as: temperature, pressure, velocity, and so on is optimized. Through the process of calculation and testing, simulation results are determined accurately and clearly. The simulation results show, when the airflow moving over the shockwave, parameters of supersonic flow changed of great value. This is the basis for identifying the movement of the airflow and optimizes the design of equipment in fact.

Keywords: *Numerical Simulation, Supersonic Flow, Shockwave, Incompressible Flow, OpenFOAM*

Introduction:

In Computational Fluid Dynamics (CFD) is great research effort has been devoted to the development of exact and efficient numerical algorithms suitable for solving flows in the various Reynolds and Mach number regimes [1]. CFD is the branch of fluid dynamics providing a cost- effective means of simulation real flows by the numerical solution of the governing equations. The governing equations for the Newtonian fluid dynamics, namely the Navier Stocks equations, have been known for over 150 years [2]. However, the development of reduced forms of these equations is still an active area of research, in particular, the turbulence closure problem of the Reynolds- averaged Navier- Stockes equations.

For non- Newtonian fluid dynamics, chemically reacting flows and two phase flows, the theoretical development is at less advanced stage. Nowadays, CFD methodologies are often employed in the fields of aircraft, turbomachinery, car, and ship design. Moreover, CFD is also applied in meteorology, oceanography, astrophysis, biology, oil recovery, and in architecture. Many numerical techniques developed for CFD are also used in the solution of the Maxwell equations or in aeroacoustics. Hence, CFD has become an important design tool in engineering, and also an indispensable research tool in various sciences [4]. Due to the advances in numerical solution methods and in the computer technology, geometrically and physically complex cases can be run even on PCs or on PC clusters. Large scale simulations of viscous flows on grids consisting of dozens of millions of elements can be completed wrong to think that CFD represents a mature technology now.

The study of high-speed supersonic flows with shock and boundary layer interaction is very importance in aerospace applications. Numerical simulations can be effectively used for parametric studies and design evaluations thus obviating the need for expensive fabrication and testing of all models [5].

A. OpenFOAM

The Open Source Field Operation and Manipulation (OpenFOAM) is a open source CFD software package. All the codes in OpenFOAM are written in C++ and it has object oriented programming interface. Besides, it provides variety of solvers both pre-processing and post-processing solvers with several finite volume solvers with structured and non-structure grids [6]. These solvers are capable of solving the steady, unsteady, compressible, incompressible, laminar and turbulent flow using FV numeric that solve a system of partial differential equations (PDE) within three dimension but can be used for one or two dimension case as well. Figure 1 depicts the case structure of OpenFOAM. OpenFOAM enables the solver structure to yield a system of equations.

The equation $\partial \rho U / \partial t + \nabla \bullet \varphi U - \nabla \bullet \mu \nabla U = -\nabla p$ is coded for example as:

```
Solve (fvm::ddt(rho, U)
+fvm::div(phi, U)
- fvm::laplacian(mu, U)
==
- fvc::grad(p)
) ;
```

The command above converts the PDE continuity equation to a sets of equation of the matrix form [P][x]=[q] where [P] consist of the coefficients extracted from the discretization process of different terms like gradient, convective, Laplacian and [x] represent the variable matrix. The matrix [q] consist of source terms.

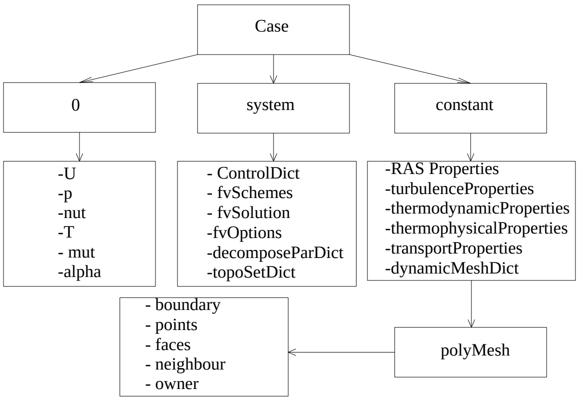

Figure 1: Case structure in OpenFOAM

OpenFOAM has been utilized as a tool for research oriented activities by numerous academic and research institutes. As it is open source software, users can customize the code for tighter tolerances. The present study is an attempt to validate the performance of OpenFOAM for predicting the shock-boundary layer interaction in a mixed compression supersonic intake. For this evaluation, the inlet configurations investigated experimentally by Schneider and Koschel [7] are considered. The predictions from OpenFOAM are compared with the experimental data reported by Schneider and Koschel [7] and also the numerical data reported by Sivakumar and Babu [8]. However, as a first step, the propagation of shock for a flow over an inclined ramp is studied with OpenFOAM and compared with the results obtained by Oliver et al [9]. Although several solvers are available in OpenFOAM for solving high speed flows, rhosonicFoam has been chosen here, since viscous effects can be modelled, contrary to the other solvers. Besides, it has the option to include turbulence models such as the k-ε, RNG (Renormalization) k–ε and realizable k-ε. Yakhot et al. [10] showed that the RNG k-ε model is appropriate for the high-speed flow under consideration. Hence, the RNG k-ε model with wall functions is used in the present study.

B. Overview of a shockwave

In physics, a shockwave or shock is a type of propagating disturbance. When a wave moves faster than the local speed of sound in a fluid it is a shock wave. Like an ordinary wave, a shockwave carries energy, and can spread through a medium; however, it is characterized by an abrupt, nearly discontinuous change in pressure, temperature and density of the medium. When a shockwave passes through matter, energy is preserved but entropy increases [11]. This change in the matter's properties manifests itself as a decrease in the energy which can be extracted as work, and as a drag force on supersonic objects; shockwaves are strongly irrverible processes.

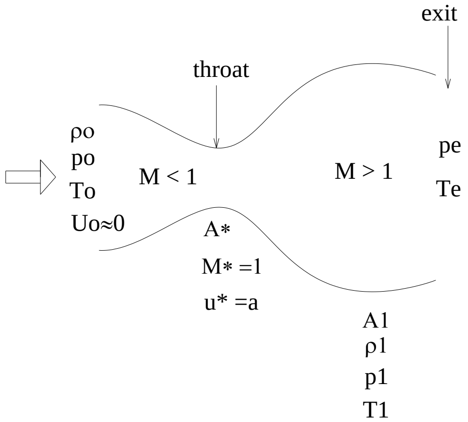

Figure 2: Supersonic flow tunnel nozzle

In fluids and solids, pressure and density changes propagate at the velocity of sound in the medium. For an ideal gas the sound velocity is:

$$a= (\gamma RT)1/2$$

Shockwaves formed about a body in a supersonic flow a created because disturbances at the body surface cannot propagate upstream since the maximum propagation speed is limited to the local speed of sound. This principle is illustrated in Fig.3, where the crosses represent disturbances traveling at a velocity U each case.

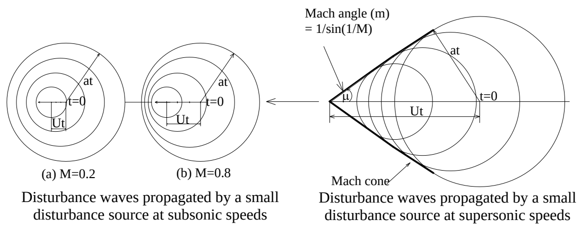

Figure 3: Evolusion of a shockwaves

C. Mach number

Mach is the most important parameter in compressible flow theory, since it compares the speed of sound in a fluid (a significant measure of compressibility effects) and the speed at which the fluid is flowing. Mach is defined as the ratio of a flow velocity to a speed of sound.

$$M=V/a$$

If Mach is defined in terms of a local speed of sound, it is called local Mach. When local Mach is used, it will be written without a subscript [12]. Mach may be defined in terms of the speed of sound at some given point in the flow, i.e., the ratio of an aircraft velocity to the speed of sound based on the ambient temperature (as opposed to local temperature). For flow in channels, ducts, and nozzles, it is sometimes more convenient to reference the Mach to a specific place in the flow. When this is done, Mach is written with subscripts or superscripts, i.e.,

$$MT =V/aT \text{ or } M^*= v/a^*$$

where aT is the speed of sound at the stagnation temperature, TT , and a* is the speed of sound at local sonic conditions.

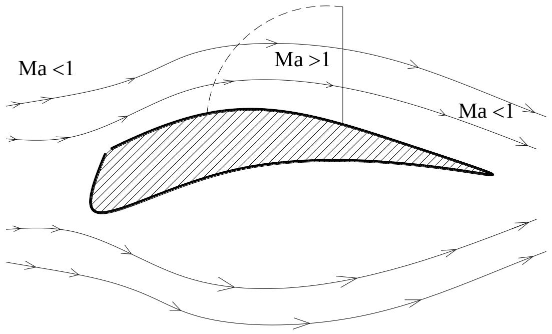

Figure 4: Mach number in transonic airflow an airfoil

Computational Methods:

A. Finite Volume Method

Finite Volume Method is used in the meshed of polyhedral cells with arbitrary number of faces each of which contains arbitrary number of vertices. The computational domain is therefore divided into several number of cell or control volumes. The number of cells can vary and there is no alignment among them generally. A cell is connected to another cell either by intersecting two cells only or the face is internal. The cells face can be treated as an external boundary. Figure 5 represent two cells or control volume (CV) of a computational domain. The first cell is connected to the second cell with a face f indicated by a surface area vector Sf. The cell which has this area vector is called owner cell and the sharing cell is called neighbor cell. Both of center of the cell is denoted by P and N respectively. Vector d represents the connecting line of the both of the center and vector dfN connects the center of the faces. The partial differential equation (PDE) are integrated over these CV's and all the volume integrals are converted into urface integrals over face f applying Gauss's theorem.

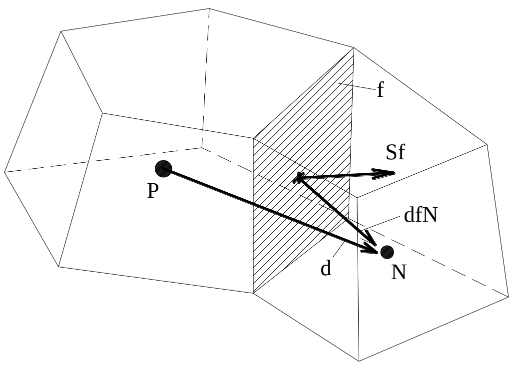

Figure 5: Finite volume discretization

In the discretization process, all the surface integrals are transformed into a set of linear algebraic equations consist of flux value of the primary variable Ψf whereas the flux values on the other faces like are found through interpolation of the flux values at the centers ΨP and ΨN respectively.

B. Typical supersonic flow problem

The gas flow moving at the speed of three times the speed of sound through a three-dimensional domain, planar geometric shape of this domain is given by the Fig.6.

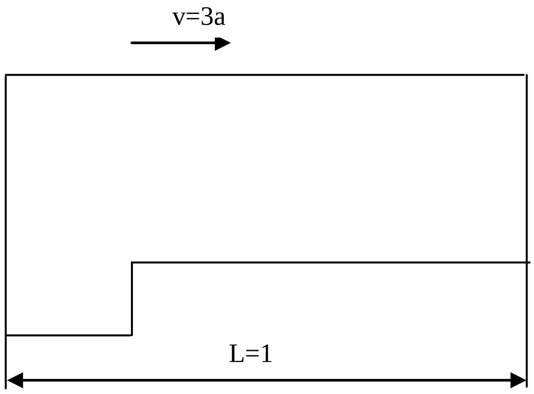

Figure 6: Typical planar geometric shape

C. Initial and boundary condition

Initial conditions are set for the time t = 0s, the speed value of air flow before shock wave surface is Ux =3m/s. Immediately, after it moves out of shock wave surface, the value of gradient U is zero. Similarly, the temperature of air flow before shock wave surface is Tx =1K. Immediately, after it moves out of shock wave surface, the value of gradient T is zero. Pressure of the air flow is declared with an initial value 1 at.

Results and Discussion:

A. Grid generation

To facilitate the observation, air flow motion is simulated 3D space domain, thereby changing the shape of the 2D grid is not required.

Figure 7: Grid simulation on OpenFOAM

The computational mesh required is created by using the OpenFOAM utility blockMesh after performance necessary edition to the file blockMeshDict. Thus, the geometry is formed based on corner points in a quadrilateral block which is meshed with hexahedral elements. The resulting structure is presented on Fig.7.

Each block comprise of twelve eight defined vertices oriented in a correct order. Also the number of cells in x-, y- and z- direction are defined in the entries. The boundary faces are defined under section patches. Names and types of patches are also defined. With the elevation of the number of mesh or grid resolution the solution fields for different variable pressure, temperature, density and velocity tends to converge to the exact analytical solution.

Present the measurements made in the experiment, compare them with preliminary work or previously published results. In the discussion section you have to relate the results to initial hypotheses.

B. Aerodynamic stream simulation

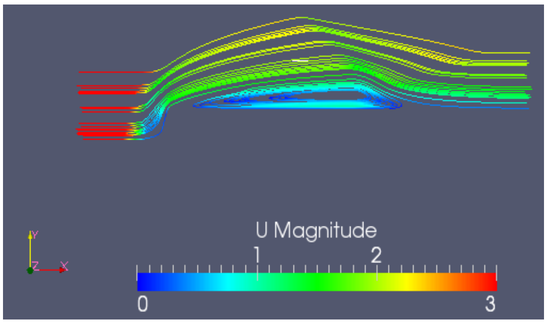

Figure 8: Aerodynamic stream simulation on OpenFOAM

Results of the simulation for aerodynamic stream proves before moving to the leading edge the velocity of the airflow achieved the greatest value (v=3), however, when out of the front edge and move to trailing edge, the airflow tends to spread with the velocity ecreasing.

C. Velocity of supersonic flow

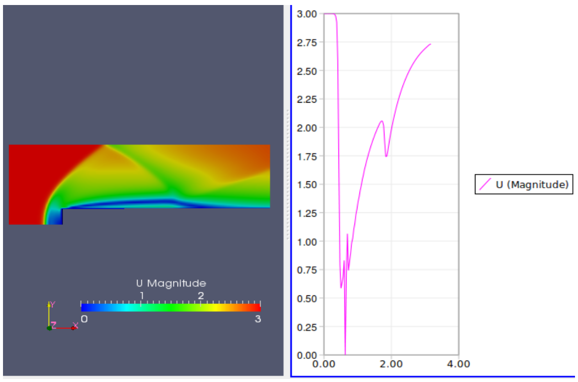

Figure 9: Relationship between of velocity (m/s) and time (s)

Fig.8 and Fig.9 shows velocity of the supersonic flow decreases dramatically when moving through shock wave surface (from 3 m/s to 0 m/s). Then, the velocity of the supersonic flow continues to rise rapidly again to the maximum value is 2.75 m / s.

D. Temperature of supersonic flow

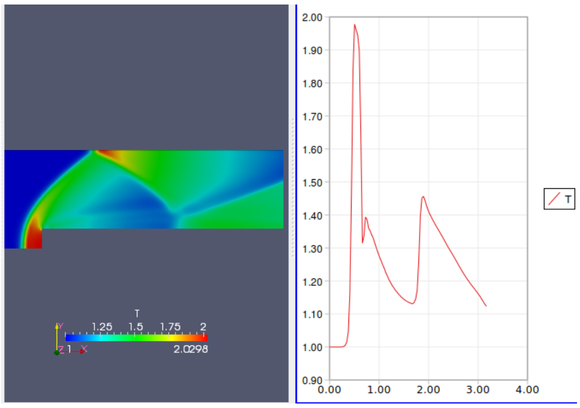

Figure 10: Relationship between of temparature (K) and time (s)

The shape of the graph the temperature variation of the supersonic flow is similar to the graph of pressure variation of supersonic flow. The temperature of the air flow does not change before shock wave surface (1K). When moving through the shock wave surface, the temperature value increased significantly and reached the maximum value at T=2K. Then it dropped and fluctuated around T = 1,25K.

E. Pressure of supersonic flow

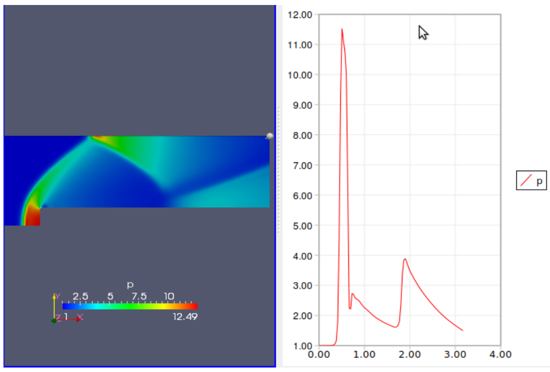

Figure 11: Relationship between of pressure (at) and time (s)

Fig.11 shows pressure of the airflow increases dramatically when moving through shock wave surface (from 1 at to 11.5 at). Then, the pressure of the airflow tends to decrease rapidly and fluctuated around 2,5at approximate value.

Conclusion and Future work:

A. Conclusion

Results achieved indicate that the velocity of the supersonic airflow decreased suddenly when moving through shock wave surface. While temperature and pressure of the supersonic flow is increased suddenly when moving through the shock wave surface. These results completely consistent with the theoretical basis.

The leading edge of the object causes a shock (left, in red) and the trailing edge of the object causes an expansion (right, in blue). It is shown in Fig.9, Fig.10 and Fig.11.

It can be stated the rhoSonicFoam shows proficiency to capture shock structure produced by supersonic air jet release. Even for a coarse grid the simulated shock structure demonstrates a clear resemblance with the basic shock structure.

The solver used here is able to predict the contact discontinuity correctly. It is explicit and unconditionally stable. All the temperature, pressure, velocity before and behind the discontinuity is not flat in the simulation as in the exact solution.

Furthermore, the solution produces bulge in higher grid resolution. The discretization scheme and the simulation time step are also the deciding factor for the magnitude and the frequency of these oscillations.

B. Future work

Although several task has been accomplished in this paper, there are lot of simulation case results yet to achieve which could be performed in following ways:

The solver rhoSonicFoam could be enhanced further by adopting non-oscillatory scheme.

More versatile cases with different set of parameters could be run.

The solver rhoSonicFoam could be enhanced further to adopt complex geometries and leak sequences.

The fluid considered for this paper work is air. The solver rhoSonicFoam could be enhanced further to provide support for multicomponent gases as well.

The solver rhoSonicFoam does not consider the ambient fluid entrainment on the actual model simulation. So, further approaches could be developed to check ambient fluid entrainment.

Acknowledgment:

The authors are indebted to College of Engineering, The University of Danang and Chinese Culture University, Taiwan for having made this work possible. The authors also appreciate the technical support of Prof. Tsing-Tshih Tsung, Department of Mechanical Engineering, Chinese Culture University, Taiwan.

References:

[1] Árpád Veress, János Molnár, József Rohács, "Compressible viscous flow solver," Compressible viscous flow solver, pp. 77–81, 2009.

[2] Wang, Yushan, "Solving incompressible Navier-Stokes equations on heterogeneous parallel architectures", Diss. Université Paris Sud-Paris XI, 2015.

[3] Sayma, Abdulnaser, "Computational fluid dynamics", Bookboon, 2009.

[4] Blazek, Jiri, " Computational fluid dynamics: principles and applications", Butterworth-Heinemann, 2015.

[5] Sudharsan, N. M., V. A. Jambekhar, and V. Babu, "A validation study of OpenFOAM using the supersonic flow in a mixed compression intake", Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering, pp 673-679,2010.

[6] OpenFOAM user Guide, OpenCFD, http://foam.sourceforge.net/doc/Guides-a4/UserGuide.pdf, United Kingdom.

[7] Koschel, W., and A. Schneider, "Detailed analysis of a mixed compression hypersonic intake", Fourteenth International Symposium on Air Breathing Engines. AIAA, 1999.

[8] Sivakumar, R., and V. Babu, "Numerical simulations of flow in a 3-D supersonic intake at high Mach numbers", Defence Science Journal, 2006.

[9] Oliver, A. Brandon, et al., Validation of High-Speed Turbulent Boundary Layer and Shock-Boundary Layer Interaction Computations with the OVERFLOW Code", American Institute of Aeronautics and Astronautics, 2006

[10] Yakhot, V. S. A. S. T. B. C. G., et al., "Development of turbulence models for shear flows by a double expansion technique" Physics of Fluids A: Fluid Dynamics, pp. 1510-1520, 1992.

[11] Emami, M. R., "Supersonic Flow and Shockwaves",2008.

[12] Ununmed, Dismlbufivfl, "Supersonic Aerodynamics," 1991.

[13] Talukdar, Mohammad., "Numerical simulation of underexpanded air jet using OpenFOAM", 2015.

[14] M. Saif Ullah Khalid, Afzaal M. Malik, "Modeling & Simulation of Supersonic Flow Using McCormack's Technique", Proceedings of the World Congress on Engineering, vol.2,2009.