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

Computation and Testing Oblique Shock over Wedge by Using OpenFOAM

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

Abstract: For the calculating and simulation, it is well known that OpenFOAM has become a very popular tool for research work in different fields and particularly, in fluid dynamics. But, it is also known its lack of detailed documentation supporting solvers made using the set of libraries provided by OpenFOAM. Among the important technical applications about oblique shock wave, calculation and testing base on OpenFOAM becomes very precise and intuitive. By using one or a combination of oblique shock waves results in more favourable post-shock conditions (lower post-shock temperature, etc.) when compared to utilizing a single normal shock. In this paper, a computational study of wedge-induced oblique shock is presented with the purpose of understanding the fundamental gas dynamics of the waves and their interactions. At two-dimensional, time accurate, finite-volume-based method was used to perform the computations. The simulation was performed with wedges of up to 37◦ semi-angle, with other inflow parameters fixed. Within the computational domain of oblique shock waves were obtained depending on the flowMach number and the wedge semi-angle. The physical mechanisms of these phenomena were analysed. This is the basis for predicting the structure more dynamic flow of reality.

Keywords: *Obilique Shock, OpenFOAM, Supersonic Flow, Shock wave, SonicFoam*

Introduction:

A. OpenFOAM

Computational fluid dynamics (CFD) has developed into an essential instrument for aerodynamics.These have concentrated on high usage of time, energy and finally the high costs associated with operating the commercial CFD based programs. Therefore, only big conglomerates utilize these commercial CFD programs. If we want to spread or create new concepts from the available commercial codes, it will entail a higher cost. So it has lead the researchers to advance their own codes or to use the existing open source codes [1]. As an open source code, OpenFOAM is used in this project and the purpose here is to compare the outcomes of the results in OpenFOAM in contrast to the well-known certified data.

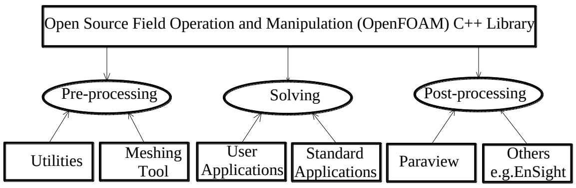

Figure 1: Overview of OpenFOAM structure

OpenFOAM is a free, open source CFD software package, licenced and distributed by the OpenFOAM Foundation [6] and developed by OpenCFD Ltd [7]. OpenFOAM is utilized in acadamia and industry to solve issuses ranging from complex fluid flows involving chemical reations, turbulence, and heat transfer, to solid dynamics and electromagnetics [8]

Nowadays, OpenFOAM has an extremely well-know tool for research work in different fields and particularly, in fluid dynamics. However, it is also known its lack of detailed documentation supporting solvers made using the set of libraries provided by OpenFOAM [2]. OpenFOAM runs in a Linux environment [3]. Unlike other tools, C++ object oriented library for numerical simulations in continum mechanics [4]. Besides, it uses the finite volume method and the SIMPLE algorithm as solution procedure [5].

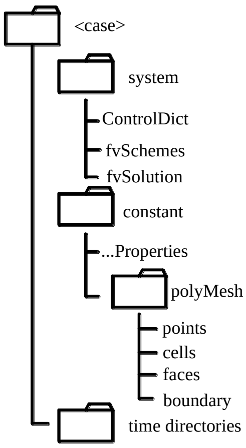

Figure 2: OpenFOAM case directory structure

An OpenFOAM simulation is characterized by a group of subdirectories, each containing specific files, as shown in Fig.2. The file structure of an OpenFOAM case is composed of a system directory, where parameters connected with the solution procedure are defined, a constant directory, which contains mesh information and physical properties for the case, and the time directories, where initial/boundary conditions and results for each recorded time step are saved [8].

B. Oblique shock

An oblique shock wave, unlike a ordinary shock, is inclined with respect to the incident upstream flow direction. It will occur when a supersonic stream encounters a corner that effectively turns the flow into itself and compresses. The upstream streamlines are uniformly deflected after the shock wave. The most common way to produce an oblique shock wave is to place a wedge into supersonic, compressible flow [9]. Similar to a normal shock wave, the oblique shock wave comprises of a very thin region across which nearly discontinuous changes in the thermodynamic properties of a gas occur. While the upstream and downstream flow directions are unchanged across a normal shock, they are distinctive for flow across an oblique shock wave.

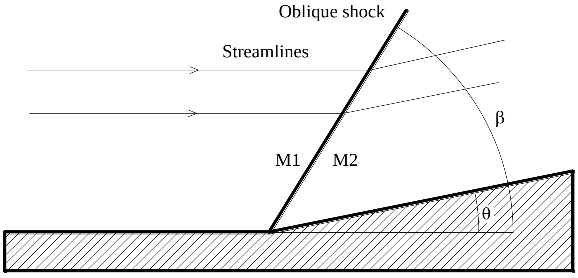

Figure 3: Supersonic flow encounters a wedge

Small disturbances created by a slender body in a supersonic flow will propagate diagonally away as Mach waves. It can be either compression waves ($p_2>p_1$) or expansion waves ($p_2 <p_1$), but in either case their strength is by definition very small ($|p_2-p_1|<<p_1$). A body of thickness, however, will generate oblique waves of finite strength. The simplest body shape for generating such waves is a concave corner, which creates an oblique shock (compression) [10]. The flow quantity changes across a normal shock.

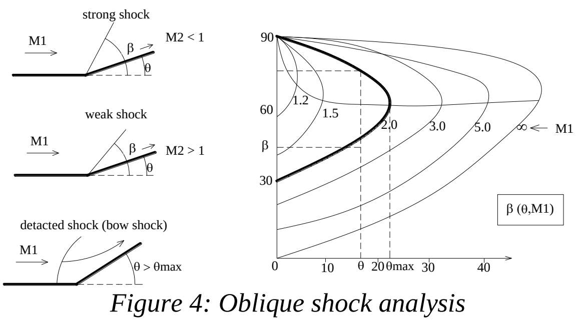

Figure 4: Oblique shock analysis

Oblique shock theory indicates that the supersonic stream of a perfect gas with a specific heat ratio of γ past a sharp wedge at zero incidence is governed by the incoming Mach number M_1 and the wedge semiangle θ. For semi-angles below a maximum value, the theory admits a weak and a strong solution. The strong solution is unrealizable physically. The weak solution comprises an oblique shock inclined at an angle β to the incoming stream. When the wedge angle equals θ_{max}, a sonic condition exists downstream of the shock [11]. When θ surpasses θ_{max}, there is no attached shock solution. Instead, a curved bow shock forms upstream of the wedge. The value of θ_{max} is a function of M_1 for a given impeccable gas. When an attached shock solution is available, there is conical symmetry in that there is no geometric lengthscale.

The shock wave may be classified as follows: normal shock wave and oblique shock wave. In this paper only focuses on oblique shock wave. An oblique shock is a sharp edged shock wave that is formed when supersonic flow is turned on itself [12]. These shocks are weaker than normal shocks, and although the temperature, pressure, density, and air stream velocity are reduced across the shock similar to the normal shock, the air stream behind the shock is not necessarily subsonic. The Mach number behind the oblique shock is calculated from the upstream Mach number, characterized by the angle at which the flow is tuned. A typical oblique shock formed by a sharp angle as shown in Fig.3

Numerical Method:

A. Finite Volume Method

Finite volume method formulation the basic idea of a FVM is to satisfy the integral form of the conservation laws to some degree of approximation for each of many adjacent control volumes which cover the domain of interest.

$$\frac{d}{dt}\int_{V(t)} \bar{U} dV + \int_{S(t)} \vec{n}\vec{F} ds = 0$$

The average value of U in a cell with volume V is

$$\bar{U} = \frac{1}{V}\iint U dV$$

$$V\frac{d}{dt}\bar{U} + \int_{S(t)} \vec{n}\vec{F} ds = 0$$

$$\frac{d\bar{U}}{dt} + \frac{1}{V}\int_{S(t)} \vec{n}\vec{F} ds = 0$$

$\bar{U}$ is the average value of U over the entire control volume, $\vec{F}$ is the flux vector and $\vec{n}$ is the unit normal to the surface. And $\vec{F} = F_I\vec{i} + G_I\vec{j}$, is the total inviscid flux, upon integrating the inviscid flux over the faces of kth control volume the above equation becomes:

$$\frac{\partial U_k}{\partial t} + \frac{1}{V_k}\left[\sum_{i=1}^{nf} \vec{F}.\vec{n} ds\right]_k = 0$$

Here,

$$\vec{n} = \frac{\Delta y_i}{\Delta s_i}\vec{i} - \frac{\Delta x_i}{\Delta s_i}\vec{j} \quad \text{and} \quad \Delta s_i = \sqrt{(\Delta x)_i^2 + (\Delta y)_i^2}$$

For 2-D axi-symmetric problems the finite volume formulation is given by:

$$\frac{d\bar{U}}{dt} + \frac{1}{V}\int_{S(t)} \vec{n}\vec{F} ds = 0$$

B. Computational Model

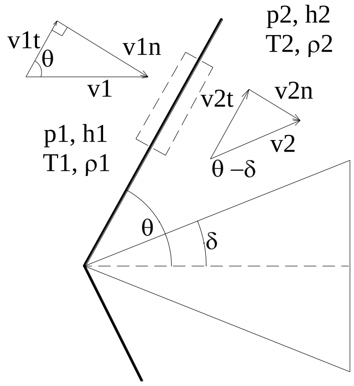

Figure 5: Oblique shock relations

$M_1 \rightarrow M_1\sin\theta = M_{1n}$
$M_2 \rightarrow M_2\sin(\theta-\delta) = M_{1n}$
Mach number (after shock)

$$M_2^2 = \left(M_1^2 + \frac{2}{\gamma-1}\right)/\left(\frac{2\gamma}{\gamma-1}M_1^2 - 1\right)$$

$$M_2^2\sin^2(\theta-\delta) = \left(M_1^2\sin^2\theta + \frac{2}{\gamma-1}\right)/\left(\frac{2\gamma}{\gamma-1}M_1^2 - 1\right)$$

Static properties

$$\frac{p_2}{p_1} = \left(\frac{2\gamma}{\gamma+1}M_1^2\sin^2\theta\right) - \left(\frac{\gamma-1}{\gamma+1}\right)$$

$$\frac{v_{1n}}{v_{2n}} = \frac{\rho_2}{\rho_1} = \frac{(\gamma+1)M_1^2\sin^2\theta}{(\gamma-1)M_1^2\sin^2\theta + 2}$$

$$\frac{T_2}{T_1} = \frac{\left(1+\frac{\gamma-1}{2}M_1^2\sin^2\theta\right)\left(\frac{2\gamma}{\gamma-1}M_1^2\sin^2\theta - 1\right)}{[M_1^2\sin^2\theta(\gamma+1)^2]/[\frac{2}{\gamma-1}]}$$

Shock angle

$$\tan\delta = \frac{\frac{2}{\tan\theta}M_1^2\sin^2\theta - 1}{M_1^2(\gamma+\cos2\theta)+2}$$

C. Typical the oblique shock problem

Air stream move over a wedge with initial velocity 650 m/s as shown in Fig.6. Deviation of velocity direction before and after the wedge is 73 degrees (θ=73°). Determine the parameters such as: T_2, v_2, p_2 and δ after the air stream move over a wedge. Knowing that T_1 =300K, $p_1 = 10^5 N/m^2$ and M_1= 2.

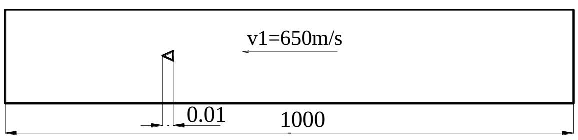

Figure 6: Typical planar geometric shape

Solution

γ=1.3, steady, adiabatic, no work, inviscid except for shock,…

$$M_2^2 = \left(2^2 + \frac{2}{1.3-1}\right)/\left(\frac{2*1.3}{1.3-1}2^2 - 1\right)$$
$$= 0.32$$

$\rightarrow M_2 = 0.56 < 1$, so subsonic after shock

$$\frac{p_2}{p_1} = \left(\frac{2*1.3}{1.3+1}2^2\sin^2 73^\circ\right) - \left(\frac{1.3-1}{1.3+1}\right)$$
$$= 4$$

$\rightarrow p_2 = 4.10^5$ N/m^2

$$\frac{T_2}{T_1} = \frac{\left(1+\frac{1.3-1}{2}2^2\sin^2 73^\circ\right)\left(\frac{2*1.3}{1.3-1}2^2\sin^2 73^\circ - 1\right)}{[2^2\sin^2 73^\circ(1.3+1)^2]/[\frac{2}{1.3-1}]}$$
$$= 1.11$$

$\rightarrow T_2 = 440K$

$$\tan\delta = \frac{\frac{2}{\tan 73^\circ}2^2\sin^2 73^\circ - 1}{2^2(1.3+\cos 2.73^\circ)+2}$$
$$= 0.32$$
$$\delta = 17^\circ$$

$v_{1n} = v_1 \sin\theta$
= 650. Sin73°
= 621 m/s

$v_{1t} = v_1 \cos\theta$
= 650.cos73°
= 190 m/s

$v_{2n} = v_2\sin(\theta-\delta)$
= 200.sin(73°-17°)
= 166 m/s

$v_{2t} = v_2\cos(\theta-\delta)$
= 200cos(73°-17°)
= 112 m/s

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.

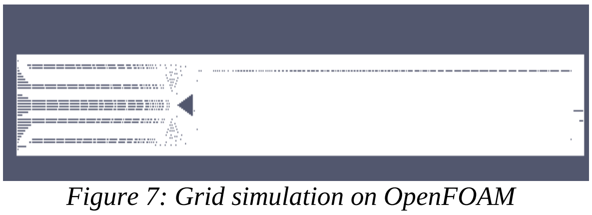

Figure 7: Grid simulation on OpenFOAM

The computational mesh required is created by using the OpenFOAM utility block Mesh after performance necessary edition to the file block MeshDict. 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.

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.

B. Aerodynamic stream

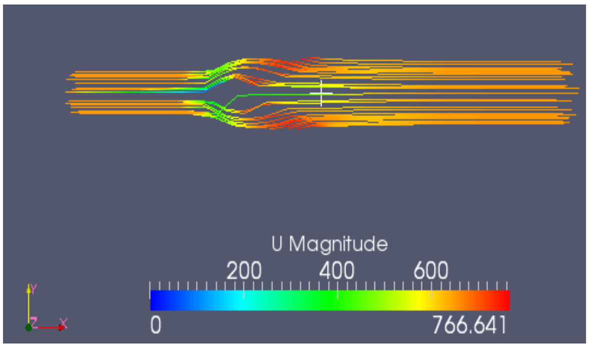

Figure 8: Aerodynamic stream simulation on OpenFOAM

C. Velocity of the oblique shock over wedge

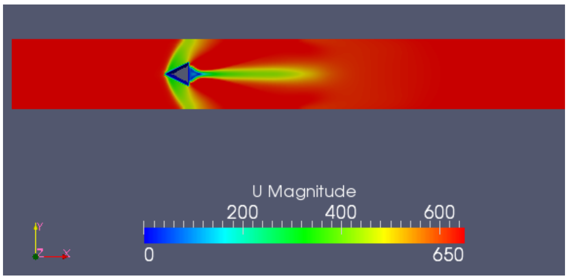

Figure 9: Velocity of the oblique shock over wedge simulation

The velocity of the airflow has a constant value (650 m/s) along the length of the flat plate. However when the airflow move over wedge, the value of its velocity is dramatically reduced. The results are shown in Figure 8 and Figure 9.

Figure 10 shows that velocity of air stream drops suddenly when it moves closer to the wedge (from 650 m/s to 380 m/s). The average value of the velocity achieved in the area around the wedge (about 550 m/s). After getting off the wedge, the velocity of the air stream back to the initial value (650 m/s).

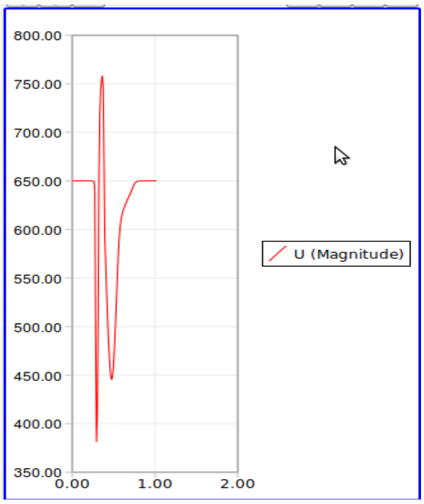

Figure 10: Relationship between of velocity (m/s) and time (s)

D. Temperature of the oblique shock over wedge

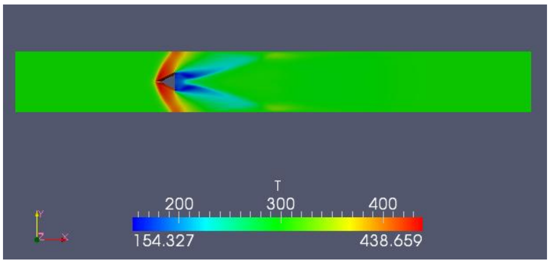

Figure 11: Temperature of the oblique shock over wedge simulation

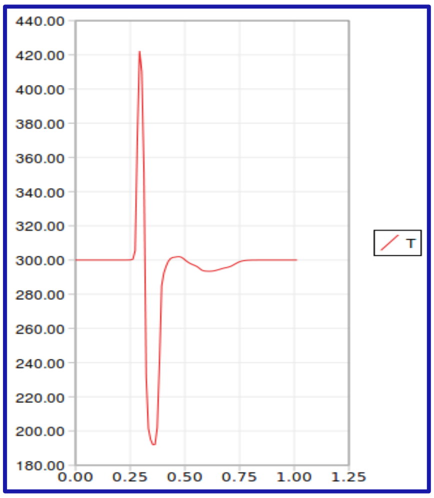

Figure 12: Relationship between of temperature (K) and time (s)

Unlike the simulation of air stream velocity, temperature of airflow along a flat gain average value (300K). The more airflow moves the wedge area nearly, the less temperature of airflow decreases. The temperature increases and reaches a maximum value of approximately 440K. Especially, this value appears in a large area around spike. After moving out of the wedge, the temperature of the air stream back the initial value. The results are shown in Fig.11-12.

E. Pressure of the oblique shock over wedge

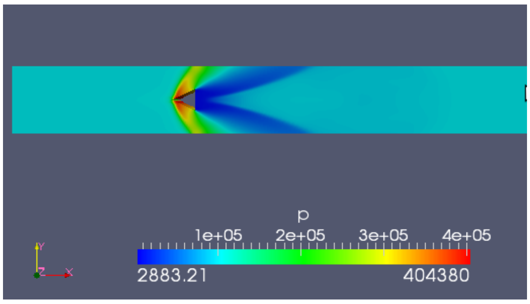

Figure 13: Pressure of the oblique shock over wedge simulation

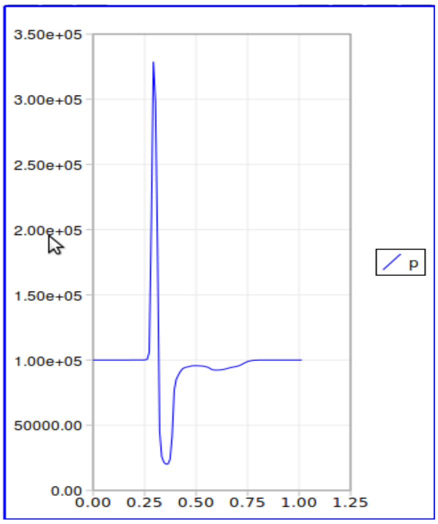

Figure 14: Relationship between of pressure (N/m^2) and time (s)

Similar to the simulation of airflow temperature, its pressure is also increasing rapidly as it moves closer to the wedge (from 10^5 N/m^2 to $3.2.10^5$ N/m^2). The maximum value of the pressure in the airflow to achieve limited narrow area at top of the wedge. After moving out of the wedge, the pressure of the air stream back the initial value. The results are shown in Fig.13-14.

Conclusion:

An unstructured finite volume solver for high speed inviscid compressible flows for 2-D and 2-D axi-symmetric configurations has been successfully developed. Results obtained for all the test cases are in good agreement with the theoretical results (oblique shock theory).

The supersonic flow through a wedge channel is a standard test case to study the oblique shock for validation of 2D inviscid flow solvers. Flow at inlet is supersonic, oblique shock will be generated at the wedge.

Results achieved indicate that the velocity of the airflow decreased suddenly when moving over a wedge. While temperature and pressure of the air stream is increased suddenly when moving over wedge. These results completely consistent with the theoretical basis.

References:

[1] UCSMHD Millewa , PM Senathilaka, WPAC Dayarathna, SA Samarasingha and SLMD Rangajeeva, (2015) "Validation of OpenFOAM as Computational Fluid Dynamics (CFD) Software", Proceedings of 8th International Research Conference, KDU, Published November 2015

[2] Marcantoni, L. F. G., Tamagno, J. P., & Elaskar, S. A., (2012) "High speed flow simulation using OpenFOAM", Asociacion Aregentina de Mecanica Computacional, Vol.31, pp 2939-2959

[3] M.J. Churchfield and P.J. Moriarty, G. Vijayakumar and J.G. Brasseur, (2010) "Wind Energy-Related Atmospheric Boundary Layer Large-Eddy Simulation Using OpenFOAM", National Renewable Energy Laboratory, Golden, CO, Report No. NREL/CP-500-48905

[4] Hrvoje Jasak, (2009) "Dynamic Mesh Handling in OpenFOAM", Proceeding of the 47th Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition, 5-8 January 2009, Orlando, FL

[5] Eric Furbo, (2010) "Evaluation of RANS turbulance models for the problems with significant impact of boundary layers"

[6] OpenCFD Ltd. (No publication date available). The open source CFD toolbox. Retrieved from http://www.openfoam.com/about/

[7] OpenFOAM Foundation. (No publication date available). Features of OpenFOAM. Retrieved from http://www.openfoam.org/features/

[8] Gomez, Sebastian, (2014) "Verification of Statistical Turbulence Models in Aerodynamic Flows" (Doctoral dissertation)

[9] Matsuo, K., Miyazato, Y., & Kim, H. D. (1999) "Shock train and pseudo-shock phenomena in internal gas flows", Progress in Aerospace Sciences, Vol.35, pp 33-100

[10] Sun, M. (1998) " Numerical and experimental studies of shock wave interaction with bodies" (Doctoral dissertation, Tohoku University)

[11] Fan, H. Y., & Lu, F. K., (2008) " Numerical modelling of oblique shock and detonation waves induced in a wedged channel", Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering, pp 687-703

[12] Shyam S. Kanwar , Aftab , Lukesh Kumar , Manoj K. Yadav , Nathu S. Singroul, (2016) "Investigation Of Oblique Shock Using FVM Based CFD Solver", International Journal for Research in Applied Science & Engineering Technology, Vol.4, January 2016, pp 278-287