Validation of OpenFOAM as Computational Fluid Dynamics (CFD) Software

UCSMHD Millewa#, PM Senathilaka, WPAC Dayarathna, SA Samarasingha
and SLMD Rangajeeva
Faculty of Engineering, General Sir John Kotelawala Defence University, Ratmalana, Sri Lanka
#millewa3907@gmail.com

Abstract— *Computational Fluid Dynamic (CFD) software is doing a critical role in the modern era of engineering. It is very important especially to the aeronautical and aerospace engineering fields. New CFD software comes in to the modern world day by day and OpenFOAM (Open source Field Operation and Manipulation) is a CFD software and, since it is open source software, it was not gone through proper validation for the results being generated. From this research project, authors aim to validate the OpenFOAM software by conducting CFD simulations for four different test cases; a 2D airfoil, finite wing, passenger car and flow over wedge. First three simulations have been performed in the subsonic regime while the latter one has been analysed for the supersonic flows. The research findings shows greater agreement between the OpenFOAM CFD predictions and validation data.* (AIAA, 1998) *Further outcomes immensely contribute to motivate researchers to utilize OpenFOAM for CFD analysis.*

Keywords— *Openfoam, Computational Fluid Dynamics, Validation*

I. INTRODUCTION

A) OpenFOAM

Computational fluid dynamics (CFD) has grown into an essential instrument for aerodynamics. These have focused on high usage of time, energy and finally the high costs associated with operating the commercial CFD based programs. Therefore, only big conglomerates use 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 progress their own codes or to use the existing open source codes. 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. (Youssef, et al., n.d.) OpenFOAM uses numerical solution procedure to get results and it offers a free license. It is also capable of handling problems as that of commercial CFD software and it has extended abilities to up-to-date, complex physics such as fluid structure interaction, complex heat or mass transfer, internal combustion engine operation and nuclear reaction. Solvers and utilities are two OpenFOAM applications, which are used to resolve particular problems and aimed to accomplish work that include data operation. Also there are other many solvers and utilities for lots of cases. The major usage of this software is that, one can generate new solvers and utilities of those who have a good understanding about computer programming, flow physics and CFD. OpenFOAM only can be installed on Linux based operating system.

B) Objectives

The primary objective of this project is to validate OpenFOAM as a CFD software by comparing known theoretical cases, experimental cases and numerical method simulation. The secondary objective is giving a basic understanding of the OpenFOAM software for new users, and enhance understanding of the OpenFOAM.

II. METHODOLOGY

A) Supersonic Flow Behaviour Over A Concave Corner

1) Detached Bow Shock Wave

In order to create a detached shock in the concave wedge; deflection angle (ϴ) and Mach number had to be chosen as to avoid solutions in ϴ-β-M diagram. So in this case, Mach no 3, and deflection angle ϴ = 45 degrees had been chosen to get a detached shock. In this case, the geometry of the wedge was designed in CATIA and exported it to GAMBIT in order to create a mesh around the wedge and to define boundaries. Then the simulations had been run on OpenFOAM. The numerical solutions had been generated as Figure 1 and Figure 2.

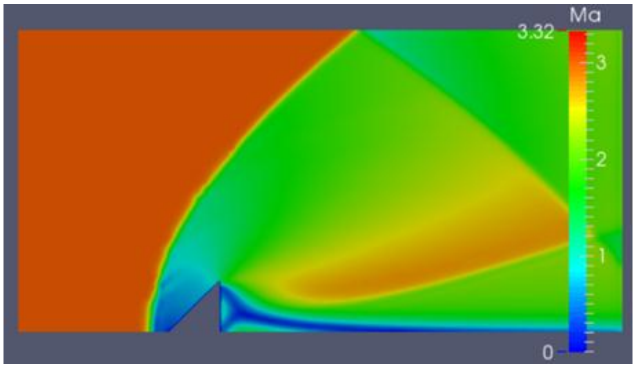

Figure 2. Mach no distribution over the detached shock wave

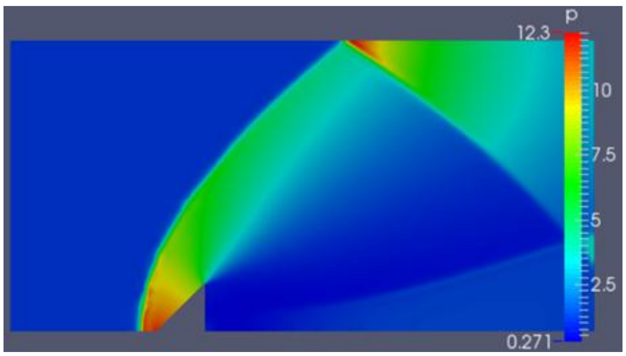

Figure 3. Pressure distribution over the detached shock wave

2) Oblique Shock Wave

The geometries in the case 2 were modelled as 2-dimensional flat plates connected to a compression corner. Here simulation type was laminar compressible flow. Pre shock pressure was 1×10^5 Pa. Ɵ is the angle (turning angle or wedge angle) between the corner and the direction of flow. β is the wave angle, the line separating the pre and post shock conditions. In this case, two geometries had been created with the angles of 15^0 and 30^0 of wedge angles. For each wedge angle, a flow with Mach number 3 and 5 had been sent over, and found respective β angles. According to the Ɵ-β-M diagram, there are relevant solutions for each case, providing oblique shock waves.

B) Aerodynamics of A 2D Airfoil – Selig S1223

The problem deals with a 2D airfoil S1223 (Figure 3) (Anon., n.d.) flying at a speed of V=38m/s at sea level and Reynolds Number is 1,010,000. Chord of the airfoil, c is equal to 1m.

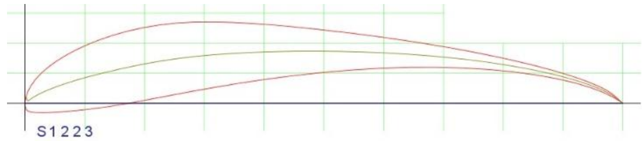

Figure 4. Selig S1223 airfoil

The creation of the mesh in this case is different from others and it helps to create mesh for any type of airfoil without causing any problems. Case is doing for 0 degree to 25 degree of angle of attacks. This simulation is explaining the 10^0 angle of attack case which can be done to other degrees of angles similarly. The case is using simpleFoam which is for steady-state solver for incompressible turbulent flow, and the Spalart-Allmaras turbulence model. (Puig, et al., June 2014) CFD numerical solution for 10^0 AoA as shown below (Figure 4 and 5) could be obtained after the simulation.

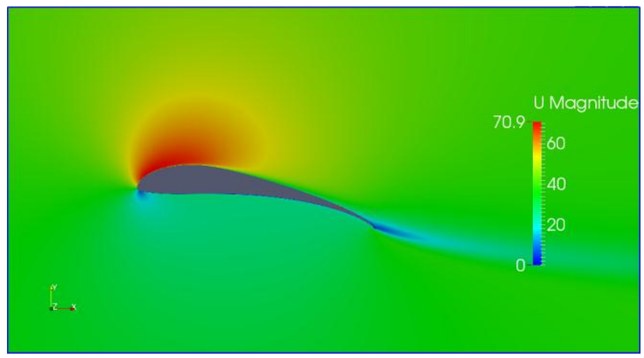

Figure 5. Velocity field around S1223 airfoil at $\alpha = 10^0$

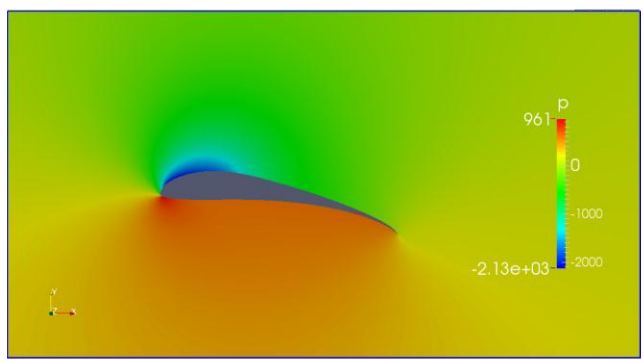

Figure 6. Pressure field around S1223 airfoil at $\alpha = 10^0$

C) Fluid Dynamics of Simple Car Model

This case discusses the simulation of simple 3D car model. The geometry of the car model is being created using an external surface, generated with CATIA V5 design software. (Legisus, 2013) Stereo Lithography (STL) file format is used to generate the geometry of the car model in OpenFOAM. The Fluid domain and the background mesh was created using blockMesh tool in

OpenFoam. Refined mesh, near to the car model geometry is created using special tool available in the OpenFoam, called as snappyHexMesh. In this case car model was driven at a velocity of 20 m/s. Flow was incompressible and turbulent flow. Hence RAS turbulence modelling was used. Sea level conditions were applied to the case. Four wheel of the car model was considered as non-moving components. k-ω SST turbulence model was implemented. After defining the boundary conditions and initial conditions the simulation was run with the required files for this OpenFOAM case.

D) Flow Over A Finite Wing

The problem encompasses 3D aerofoil NACA 0018 *(Anon., n.d.)* At a speed of V = 45 m/s at Sea level for a three different angle of attacks were simulated. The mean aerodynamic chord is the chord of a rectangular wing which has the same area, aerodynamic force and position of the pressure center for a given angle of attack as the original.

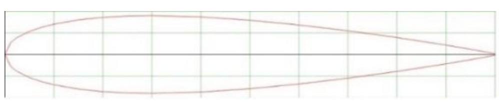

Figure 7. NACA 0018 airfoil

All steps of mesh generating was done in the OpenFOAM itself. After carried out the simulation following numerical solutions were obtained for 0^0 AoA as Figure 7 and 8.

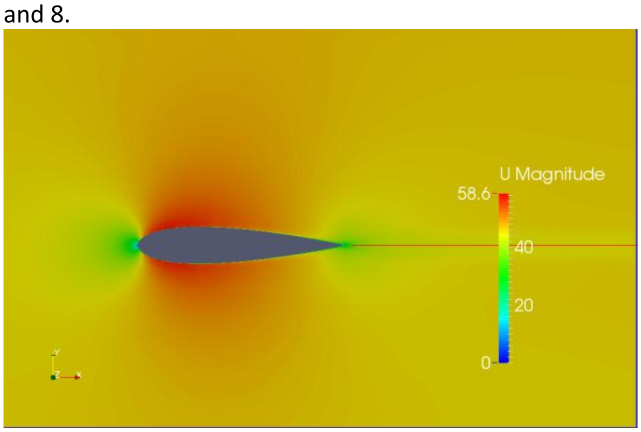

Figure 8. Velocity variation in whole domain

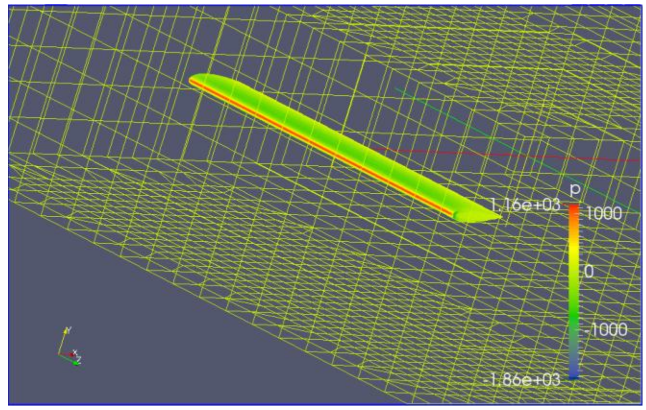

Figure 9. Pressure Variation around 3D wing

III. DATA INTERPRETATION AND ANALYSIS

A) Supersonic Flow Behaviour Over A Concave Corner

The numerical simulation visualizes the physics by presenting the images of the distribution of the flow properties, such as the Mach number, temperature, and pressure. And numerical solutions always show ranges of results because it shows the results at many points in the boundary region. Analytical approach provides exact solutions, but only between two chosen points. Here, in the case "supersonic flow over wedge", the comparisons had been done between analytical and numerical solutions in order to verify the numerical results. (Walker & Schmisseur, n.d.) As in the analytical solutions (in the first case - detached bow shock), the downstream pressure and Mach number at a point just after the normal shock wave was 10.33×105 Pa and 0.4752 respectively.

And, that in numerical results varied between 10 to 12 (pressure) and 0 to 0.5 (Mach) respectively. The table 1 compares the analytical and numerical solutions in oblique shock. When the Mach number increases, the wave angle decreases. In these cases, the higher Mach number results in the pressure to rise and also the temperature to rise behind the shock wave; and a higher wedge angle results in the wave angle to increase and also the pressure and temperature to increase behind the shock wave. And when studying the analytical solutions which had been solved during the simulations of the case "supersonic flow over wedge", the results were almost alike as in the numerical solutions. So the OpenFOAM CFD software can be verified according to the first case, since it has provided accurate solutions in supersonic cases.

B) Aerodynamics of A 2D Airfoil – Selig S1223

For Data interpretation and analysis following graphs were generated for CFD solution as Figures 9, 10, 11 and 12 and then numerical data was compared with analytical data. (Tejni, December 1996)

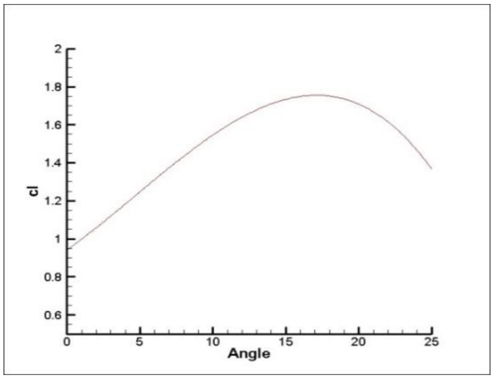

Figure 10. C_l vs. alpha; CFD solution

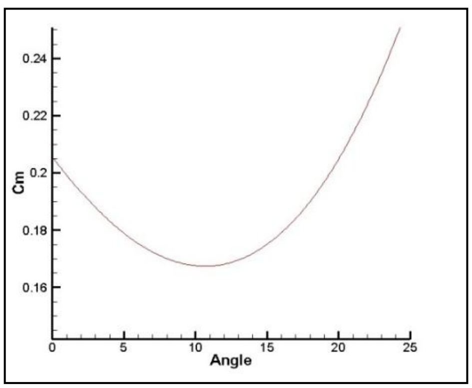

Figure 10. C_m vs. alpha; CFD

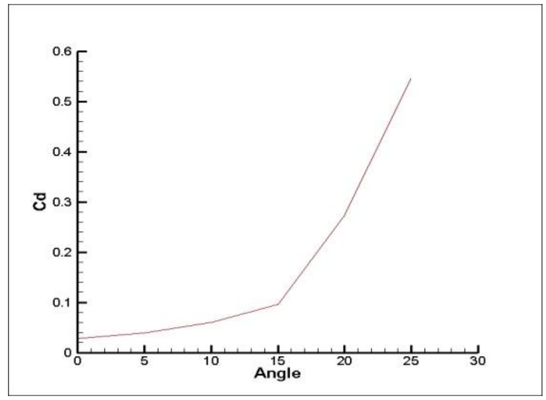

Figure 11. C_D vs. alpha; CFD solution

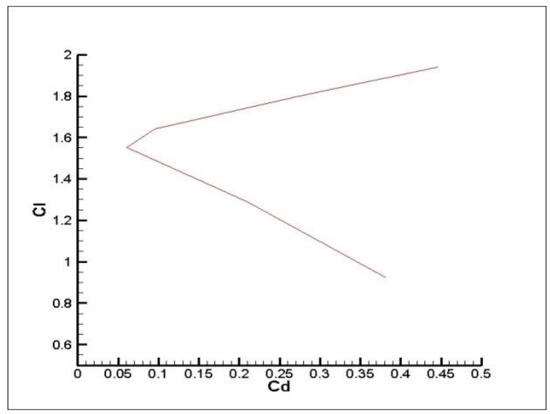

Figure 12. C_l vs. C_D; CFD Solution

Table 3 : Comparison of analytical and numerical data in oblique shock wave

Upstream Mach No (M_1)	Wedge angle (Θ)	Shock angle (β)		Downstream pressure (P_2)		Downstream Mach No (M_2)	
		analytical	numerical	Analytical $\times 10^5$ Pa	Numerical $\times 10^5$ Pa	analytical	numerical
3	15^0	32.2404^0	33^0	2.82	2.49	2.255	2.25
5	15^0	24.3217^0	25^0	4.78082	4.08	3.50405	3.56
3	30^0	52.0136^0	52^0	6.35585	5.4	1.40594	1.89
5	30^0	42.3442^0	43^0	13.0666	11.5	2.13564	2.57

In C_l vs. AoA, for analytical and experimental it has $C_{l,max}$ = 2.425. But in CFD solution it gives as 1.799.

C_m vs. AoA, there were uncertainties about the point of moment took in analytical and experimental data (Marikkar, et al., 2014), validation couldn't be done properly even though it showed a same behaviour pattern in graphs.

It gave approximately similar behaviour and values for the positive side of the C_D vs. AoA since the simulation had done only for the positive AOAs. C_l vs. C_D gave false answer even though with corrections and re-simulations.

C) Fluid Dynamics of Simple Car Model

At the end of the simulation to generate the Figures 13 and 14, VTK plane which is included in the postProcessing directory is used. Figure 13 shows the pressure distribution around the car. Since the car model has conventional design more drag is created at the nose as well as the windscreen. It can be identified by the pressure increments on those areas. Most of the car designs in future are designed considering these facts. The new car models are designed to avoid these high drags. Square shape nose creates more drag than streamline shaped nose which can be seen in modern car models. (Shinde, et al., n.d.)

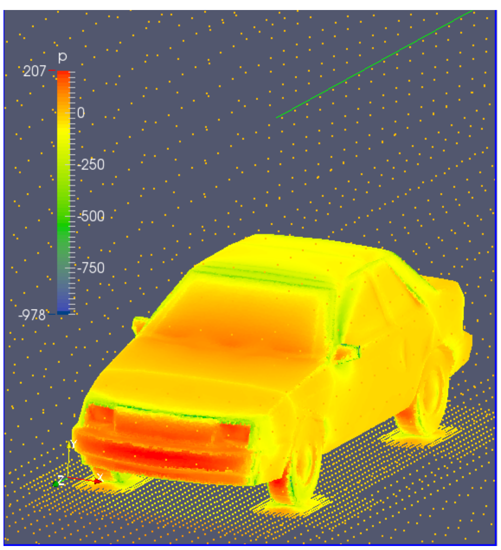

Figure 13. Pressure Field around the car

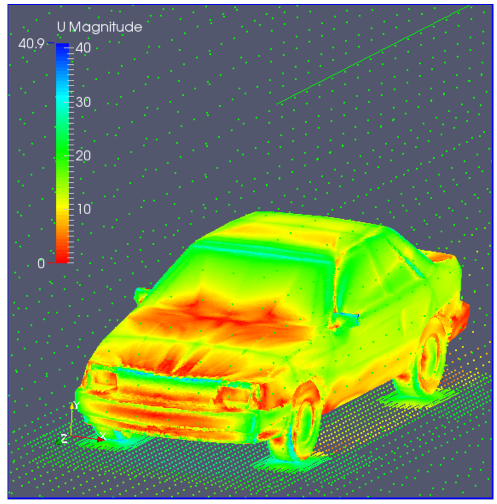

Figure 14. Velocity Field around the car

The Figure 14 shows the velocity field around the car surface. The stagnation areas can be identified as the nose and the wind screen. High intensity of the red color shows the low velocity areas where the magnitude of the velocity is almost equal to the zero.

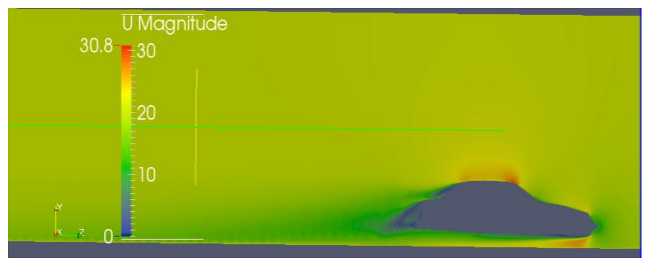

Figure 15. The velocity field around the whole domain

Figure 15 shows the velocity field in the whole domain. At the nose of the car the drop of the velocity is identified by the green color. In this figure more turbulence is visible at the back of the car. It creates uncomfortable conditions for the persons inside the car. Although the case is set turbulent, the wake of the car shows a continuous appearance. It is because a RAS turbulence model has been used. RAS turbulence model is based on the average flow conditions. If a turbulence model such as LES or DNS, it would have been possible to observe turbulence in the wake.

After doing the simulation the results says that the car model design is not having perfect aerodynamic performance. More turbulence is creating at the back. Most of the old car models are not designed considering the aerodynamic performance. But today, not only the sports car but also the normal car models are created considering aerodynamic performance. Since these kind of CFD software are available, the alternations can be done without making several car models. Only thing have to do is alter the design according to the requirements. Hence modern car designers used CFD software to optimize their design performance.

D) Flow Over A Finite Wing

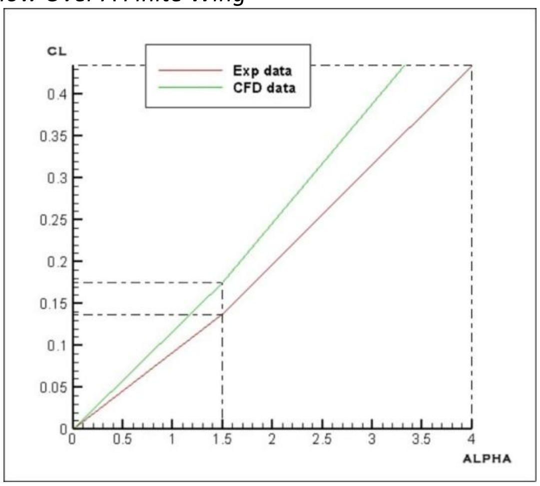

Figure 16. Coefficient of lift vs. angle of attack curve

At the zero angle of attack the lift in Figure 16 which is created by the aerofoil is zero since the aerofoil is symmetric. At the 1.5^0 of AoA, C_L is having experimental value of 0.1373. Value of 0.1756 for C_L is given by the CFD. The deference between the two values is 0.0383. At the 4^0 of AoA, C_L is having experimental value of 0.4247. Value of 0.5314 for C_L is given by the CFD. The deference between the two values is 0.0967.

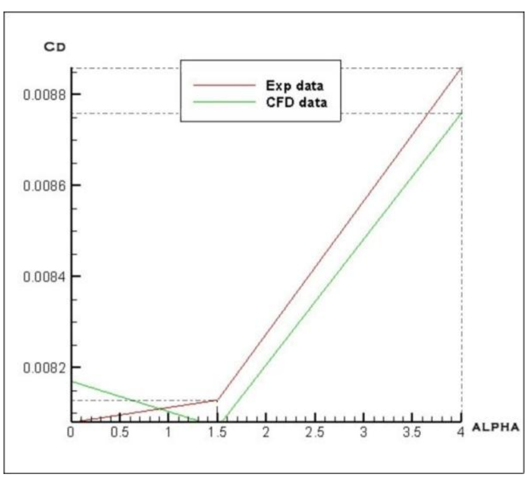

Figure 17. Coefficient of drag vs. angle of attack curve

At the 0^0 of AoA as Figure 17 C_D is having experimental value of 0.00808. Value of 0.00817 for C_D is given by the CFD. The difference between the two values is 9×10^{-5}. At the 1.5^0 of AoA, C_D is having experimental value (Anon., n.d.) of 0.00813. Value of 0.00807for C_D is given by the CFD. The difference between the two values is 6×10^{-5}. At the 4^0 of AoA, C_D is having experimental value of 0.00886. Value of 0.00876for C_D is given by the CFD. The difference between the two values is 10×10^{-5}.

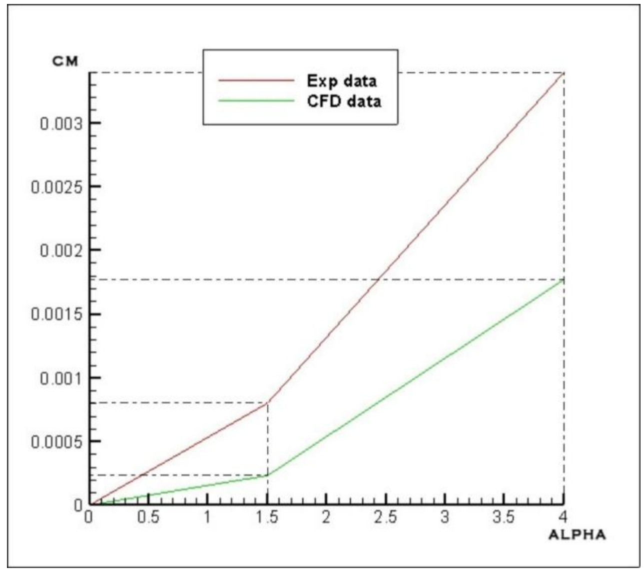

Figure 18. Moment coefficient vs. angle of attack curve

Table 4 : Moment coefficient comparison table

Angle of attack	Experimental data	CFD data	Deviation
0	0	0	0
1.5	0.0008	0.00233	1.53×10^{-3}
4	0.0034	0.00177	1.63×10^{-3}

IV. CONCLUSION

Here, four cases were studied and simulated and the results were compared with the available experimental data and solved analytical data to verify and validate the software. In the first case, the flow properties relating to the shock waves were impossible to gain through the experiments due to the insufficient resources. Hence, it was compared only with solved analytical data, which was preferably verified with numerical results.

Numerical solutions in other three cases were compared with experimental data, and in some areas, there were some considerable deviations between the results too. These deviations are due to the errors of which are arising during the CFD simulation. They have the tendency to accumulate through computational processes that may yield unrealistic CFD solutions. Physical modelling error was prone in the case 3 “flow aver a car model" due to the degree of uncertainty in the designed model parameters with the real phenomenon and unavailability of experimental data to compare with the designed model.

Analysing aerodynamic of 2D airfoil was become difficult due to the uncertainty of data that use to generate analytical and experimental solution. Especially the Reynolds' number showed lot more different value of than what should actually happened in problem condition

It does not necessarily imply that OpenFOAM is validated and verified correctly, since in this project only 4 cases had been used for verification and validation. Also only a narrow area is considered throughout this project whereas CFD can be used in a wide range of applications.

REFERENCES

AIAA, 1998. Guide for the Verification and Validation of Computational Fluid Dynamics Simulations. *AIAA G-077-1998.*

Alhussan, K., 2013. Computational analysis of high speed flow over a conical surface with changing the angle of attack. *Procedia Engineering,* Volume 61, pp. 48-51.

Anon., n.d. *CFD Online.* [Online] Available at: http://www.cfd-online.com/Forums/openfoam-verification-validation/ [Accessed 04 04 2015].

Anon., n.d. *HPC Wiki.* [Online] Available at: https://www.hpc.ntnu.no/display/hpc/OpenFOAM+-+Airfoil+Calculations [Accessed 20 10 2014].

Anon., n.d. *UIUC Applied Applied Aerodynamics.* [Online] Available at: http://m-selig.ae.illinois.edu/ads/coord_database.html [Accessed 10 08 2014].

Hall, N., n.d. *Glenn Research Center.* [Online] Available at: https://www.grc.nasa.gov/www/k-12/airplane/wdgflow.html [Accessed 10 03 2015].

Legisus, 2013. *3DVIA.* [Online] Available at: http://www.3dvia.com/models/72F6C6687A4C5E70/car [Accessed 23 12 2014].

Marikkar, S. et al., 2014. *Department of Aeronautical Engineering, General Sir John Kotelawala Defence University.* [Online] Available at: http://www.kdu.ac.lk/department-of-aeronautical-engineering/images/projects/mini_projects/BASIC-AIRFRAME-SYSTEM.pdf [Accessed 27 12 2014].

Oberkampf, W. & T., G. T., 2000. *Validation MEthodology in Computaional Fluid Dynamics,* Denver, CO: AIAA.

Puig, J. C., Perdo, J. G. & Gustavo, R., June 2014. *OpenFOAM GUIDE FOR BEGINNERS,* Barcelona: The Foam House.

Shinde, G., Joshi, A. & Nikam, K., n.d. *Numerical_Investigations_of_the_Driver_Car_Model_using_Opensource_CFD_Solver_OpenFOAM,* s.l.: s.n.

Slater, J. W., 2008. *Glenn Research Center.* [Online] Available at: http://www.grc.nasa.gov/WWW/wind/valid/tutorial/tutorial.html [Accessed 03 - 05 2015].

Tejni, E., December 1996. *Computaional Investigation of the Low Speed S1223 Airfoil with and without a gurney flap,* Ann Arbor, MI: UMI Microform.

Walker, S. & Schmisseur, J. D., n.d. *CFD Validation of Shock-Shock Interation Flow Fields,* Arlington, VA: USA Air Force Research Laboratory.

Youssef, A., Aimad, .-R. & Maxime, G., n.d. *OpenFOAM Run-Time Post Processing.* [Online] Available at: http://hmf.enseeiht.fr/travaux/projnum/book/export/html/901 [Accessed 28 04 2015].

BIOGRAPHY OF AUTHORS

HD Millewa, is a final year undergraduate at theDepartment of Aeronautical Engineering in General Sir John Kotelawala Defence University

PM Senathilaka is a final year undergraduate at theDepartment of Aeronautical Engineering in General Sir John Kotelawala Defence University

WPAC Dayarathna is a final year undergraduate at theDepartment of Aeronautical Engineering in General Sir John Kotelawala Defence University