CFD ANALYSIS OF ONERAM6 WING USING OPENFOAM

SH Subahan#, GDAL Gohumulla, PDKL Sriyarathne, WMSKB Wijayathunga, SLMDR Samarathunga and DJK Lokupathirage

Department of Aeronautical Engineering, Faculty of Engineering, General Sir John Kotelawala Defence University, Sri Lanka.

shssharshas@gmail.com

Abstract— Computational Fluid Dynamic (CFD) is very important in the aviation industry. Especially it is critically effecting to the model analysis of aeronautical and Aerospace engineering filed. Nowadays lots of CFD software on the market. The main object of these project to validate OpenFOAM®, one of the leading CFD software in the world. The simulation was based on 0.8395 Mach number with Reynolds Number of 11.72 x 10^6 of air flow parameters over ONERA M6 wing section at 3.06 degrees angle of attack. The ONERA M6 wing especially using for the wind tunnel experiment since 1972 by NASA. SolidWorks model of ONERA M6 was used to generate computational mesh and CFD simulations were done with OpenFOAM. Finally by using simulated Coefficient of Pressure (Cp) data values and wind tunnel data values are compared for the validation.

Keywords— *OpenFoam, Computational Fluid Dynamics, Validation, ONERA M6*

I. INTRODUCTION

Computational Fluid Dynamics (CFD) methods are being developed as an alternative to wind tunnel experiments, to replace the real experiments by the numerical experiments. Compared to wind tunnel experiments, CFD methods are less expensive and require less time. The accuracy of the CFD solutions depends upon the complexity of the physical and numerical modeling utilized.

CFD is the process of using computers to simulate realistic flows. CFD has become a powerful tool in the analysis and understanding of any type of flow phenomena such as in-viscid/viscous, compressible/incompressible, heat and mass transfer, phase change and many more, about any kind of geometry, such as aircraft, automobile, ship, etc. For the simulation the type of model should dependent on the accuracy needed, the computer power accessible and the time scale to accomplish the analysis. The strategy of CFD is to replace the continuous problem domain with a discrete domain using a grid. In a CFD solution, one would directly solve for the relevant flow variables only at the grid points. The values at other locations are determined by interpolating the values at the grid points. This method is based on the Navier-Stokes equations. These equations describe how the velocity, pressure, temperature, and density of a moving fluid are related.

A brief description on the software we used for our project to get through our tasks. OpenFOAM is a free-to-use Open Source numerical simulation software with extensive CFD and multi-physics capabilities and created as a C++ library, used to create executable, known as applications. The applications fall into two categories: solvers, that are each designed to solve a specific problem in continuum mechanics; and utilities, that are designed to perform tasks that involve data manipulation. The OpenFOAM distribution contains numerous solvers and utilities

Covering a wide range of problems. One of the strengths of OpenFOAM is that new solvers and utilities can be created by its users with some pre-requisite knowledge of the underlying method, physics and programming techniques involved. OpenFOAM is supplied with pre- and post-processing environments. The interface to the pre- and post-processing are themselves OpenFOAM utilities, thereby ensuring consistent data handling across all environments.

The M6 arrow shaped wing was designed by Bernard Monnerie and his aerodynamicist colleagues at Onera in 1972, to serve as experimental support in studies of three-dimensional flows at transonic speeds and high Reynolds numbers (conditions representative of the actual flight of military and civilian aircraft). This wing is original due to having been defined in a purely analytical way. The ONERA M6 wing is a swept, semi-span wing with no twist. It uses a symmetric airfoil using the ONERA D section.

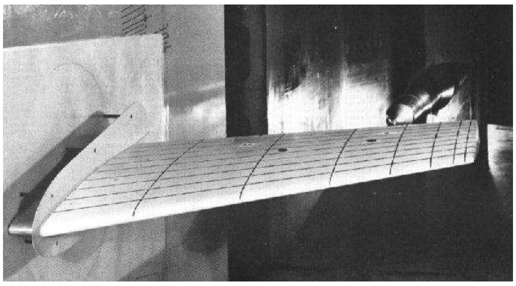


Figure 1 Onera M6 wing

The Onera M6 wing is a classic CFD validation case for external flows because of its simple geometry combined with complexities of transonic flow.

Table 1: Onera M6 wing specification

Wing Plane form	Swept Back Wing
Aspect Ratio	3.8
Leading Edge Sweep	30
Trailing edge sweep	15.8
Taper ratio	.562
Mean aerodynamic chord	c=0.64607m (without twist)
Span or semi span	b = 1.1963 m
Sweep Angle	26.7

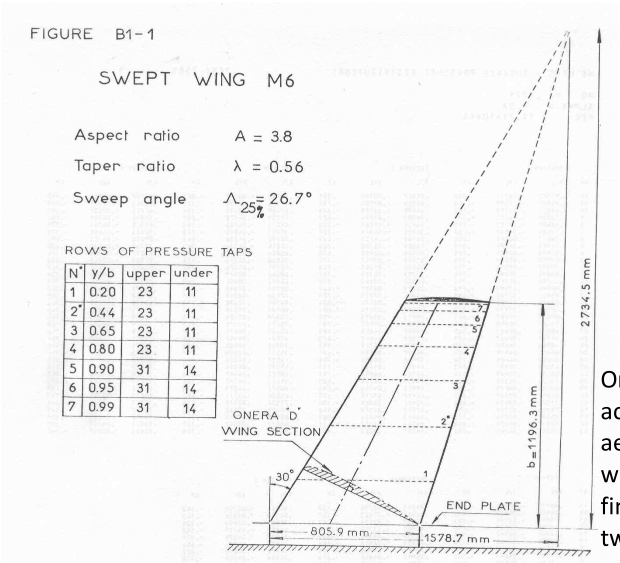


Figure 2: Onera M6 specifications

SolidWorks is a solid modelling computer-aided design (CAD) and computer-aided engineering (CAE) computer program that runs on Microsoft Windows. It is the software which we used to model the ONERA M6 wing. SolidWorks is published by Dassault Systems. SolidWorks Corporation was founded in December 1993 by Massachusetts Institute with the goal of building 3D CAD software that was easy-to-use, affordable, and available on the Windows desktop. SolidWorks released its first product SolidWorks 95, in November 1995.

The following objectives has been set-up in order to achieve the final outcome.

1. Making Onera M6 wing on SolidWorks software and performing Computerized Fluid Dynamics (CFD) analysis with OpenFOAM.
2. CFD results are compared with wind tunnel data and validate the openFOAM

II. METHODOLOGY

A. *Modelling*

Oneram6 aerofoil coordinates were downloaded from the NASA web page link. It provided only the upper surface coordinates, in order to complete the aerofoil shape excel was used. Since oneram6 is a symmetry aerofoil upper surface Y coordinates were rearranged accordingly with X coordinates along the chord and Z values were set as 0, as we are creating a 2D aerofoil.

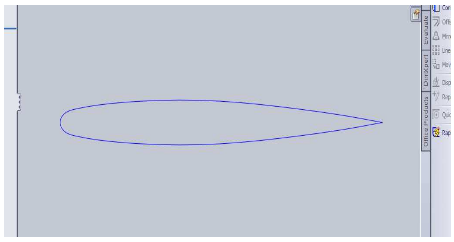


Figure 3: Onera M6 2D aerofoil

B. *Creating the wing*

Orientation was set to top and a sketch was made according to the wing geometry such as Mean aerodynamic chord, Span etc. Next the aerofoil shape was made at the tip and scaled it accordingly. To finally make the wing shape was lofted between the two aerofoils. For the validation purpose we have to maintain 3.06 degree angle of attack. To obtain the

3.06^0 angle of attack, wing was rotated around the Z axis.

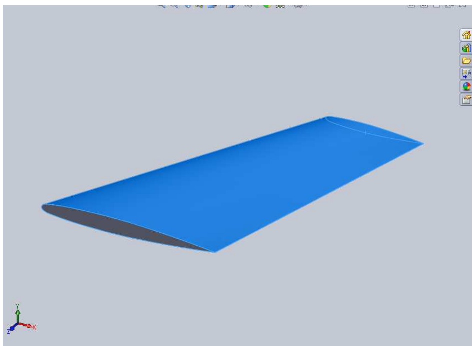


Figure 4: Onera wing solid Model

C. Meshing

Second process in pre-processing was to develop a mesh. In order to obtain valid and accurate results creating a fine mesh is important. Mesh was completed using OpenFOAM with the use of “snappyHexMesh” mesh generator. Before creating the wing mesh the domain mesh was successfully created with the use of command “blockMesh”. And the “surfaceFeatureExtract” command was given to create an emesh file and then “snappyHexMesh” was given. But it was not a refined mesh. So again some steps were initiated to refine the mesh.

Modifications were created to define a refinement box which the first was covering whole wing, and the areas where the flow is having its critical variations. But to accommodate a mesh with more accuracy 3 layers were developed near to the wing surface.

The solidworks geometry of the wing was then saved in the format of “stl” as it is the compatible with OpenFOAM. Then this stl file was named as “oneram6.stl”.The case directory was named as oneram6 case and the folders were contained with the files including the relevant information to simulate this case. This case was simulated using simpleFoam and SST k-ω turbulence model at a Mach number of 0.8.

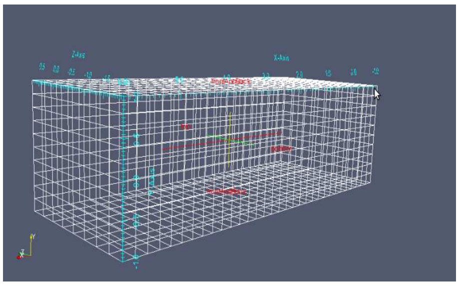


Figure 6: Domain Mesh

In refinementSurfaces dictionary in castellatedMeshControls requires dictionary entries for each STL surface and a default level specification of the minimum and maximum refinement in the form (<min> <max>). In this research it was valued min (-0.1 -0.05 -0.025) max (0.4 0.075 0.025).

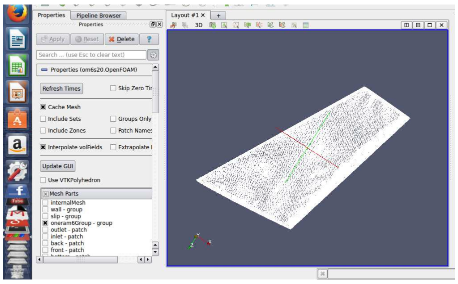


Figure 7: Onera M6 wing Mesh

D. Simulation parameters

Mach number: 0.8395

Angle of Attack: 3.06 degrees

Reynolds Number: 11.72×10^6.

Stagnation temperature 283 Kelvin (K)

E. Initial and boundary condition

The initial and boundary conditions are situated in the zero time folder as mentioned before in the case folder structure.

Table 2: Initial Conditions

flow Velocity	(285.4 0 0)
pressure	0
Turbulent KE	0.24
Turbulent Omega	1.78
#input Mode	merge

For each variable the condition is given for all the patches created by the blockMesh and the addition of the Wing from the snappyHexMesh. The tables below give the boundary conditions for each variable and for each type of case. Some annotations are as follows, FP is free stream pressure, FV is fixed value, ZG is zero gradient and IO is InletOutlet. The numbers in the brackets are the uniform value assigned for the field at that boundary.

For most boundaries they have the type patch which means it doesn't have geometric information for the mesh. The wing is a wall type which allows wall functions to be assigned for the turbulence model and others are symmetry plane which means that values on both sides of the plane are equal. Zero gradient means that the field at that boundary has a zero normal gradient. inletOutlet means the flow is mixed depending on the direction of the velocity. When the flow is in then the value is fixed and when it is out the value is zero gradient.

Table 3: Booundary Conditions

Boundary	type	U	p	k	nut	omega
Inlet	Patch	free Stram	FP	FV	calculated	FV
Outlet	Patch	free Stream	FP	IO	Calculated	FV
Top	Slip	Slip	Slip	Slip	Calculated	Slip
Bottom	Slip	Slip	Slip	Slip	Calculated	Slip
FrontAndBack	Slip	Slip	Slip	slip	calculated	Slip
Oneram6	Wall	FV	ZG	kqR Wall Function	Nutk Wall Function	Omega Wall Function

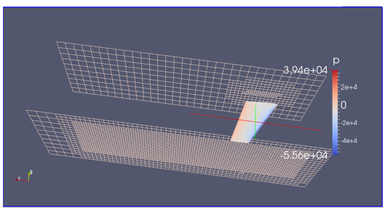


Figure 8: Front and back boundary

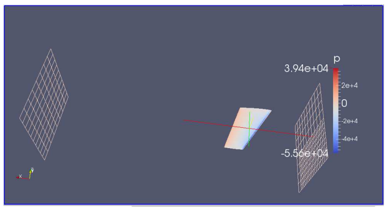


Figure 9: Inlet and Outlet boundary

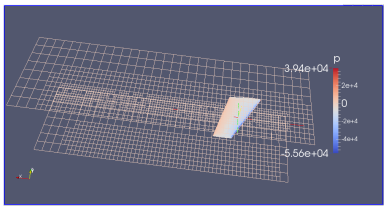


Figure 10: Top and Bottom boundary

F. *Turbulence modelling*

SST k- ω turbulence model is a two equation eddy viscosity model. It is a combination of a *k- ω* model (In the inner boundary layer) and the *k- Ɛ* model (In the outer region of boundary layer). This includes two transport equations for the turbulent properties of the flow which one variable is turbulent kinetic energy (*k*) to determine the energy in the turbulence and the other variable is the specific dissipation (ω) to determine the scale of the turbulence with respect to the Equation and Equation. The *SST K- ω* formulation can be used in the inner parts of the boundary layer even at low-Re turbulence model without any extra damping functions. Moreover this model is excellent in performing under condition of separating flow. But it has a limitation of the shear stress in adverse pressure gradient regions.

III. RESULTS AND DISSCUSSION

The information relating to the control of the solution procedure are read in from the controlDict dictionary. For this case, the startTime is 0. In this situation it is best to set the time step deltaT to endTime. So it simply acts as an iteration counter for this case.

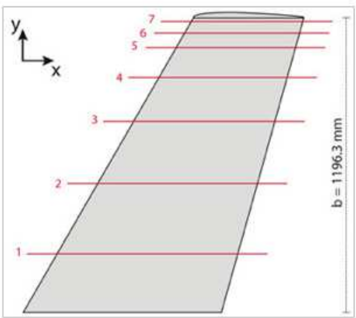
As for visual comparisons, the selected span sectional locations were based on the available data from wind tunnel test results. Post simulation, the surface pressure data at those wingspan locations were sampled .which data are sampling and producing the output Cp values were done in ParaView. It was achieved by the usage" Plot over line‟ filters.

Velocity variation with the angle of attack was conducted 3.06, 15, and 20 degrees because the flow separation to be captured and for the validation part at last.

3.06 angle of attack was not resulting much variation of the flow field. But angle of attack is increased there are identified large amount of variation can be found. When it is comes to the 20 degrees of AOA it seems flow separation was done. So we can found between the 15 and 20 degrees of AOA the flow separation was happened.

For the accurate and reliable validation, Onera M6 wing is divided into main seven parts along the span wise direction.

20%, 44%, 65%, 80%, 90%, 95% and 99% of (y/b) sections are introduced by Schmitt and Charpin.



Section	y/b*
1	0.20
2	0.44
3	0.65
4	0.80
5	0.90
6	0.95
7	0.99

*b is the wing span.

Figure 11: Span wise Locations on Onera M6 wing

First of all for the validation, all the graphs are drawn by using pressure coefficient value which is given by Charpin experiment. According to the (y/c) value, there are seven separate graphs which is included pressure coefficient of both upper and lower surface.

Each and every section is included 11 pressure taps in lower surface and 23 pressure taps in upper surface. Though we planned to find the pressure coefficient value of each point where the pressure taps were located, because of the limitations of time and lack of computer facilities, finally we were got through the rough validation of the openFOAM. Because openFOAM software is not given pressure coefficient values directly. They are given pressure values of each point, then pressure coefficient value should be calculated by using bellow equation (RAhman, 2015).

Here we have drawn all the pressure coefficient graphs according to the pressure tap located in (y/C) planes. And the same time simulated pressure graphs also shown below under (y/C).

All the results are collected 3.06 degrees of angle of attack because experimental data which we collected was a wing which was angled 3.06 degrees.

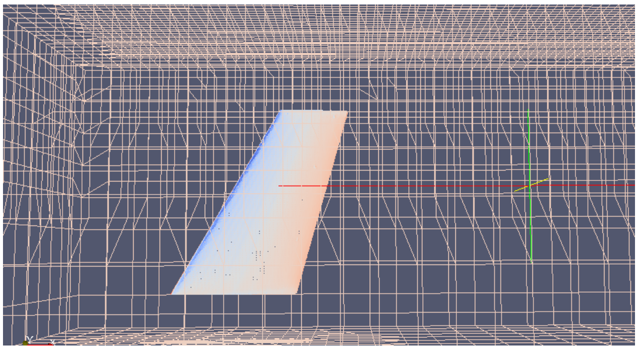


Figure 12: U variation 3.06 degree AOA (Top View)

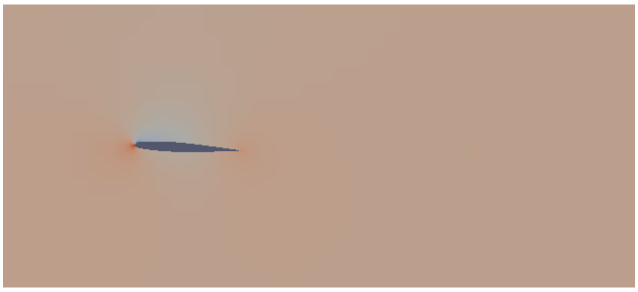


Figure 13: U variation 3.06 degree AOA (side view)

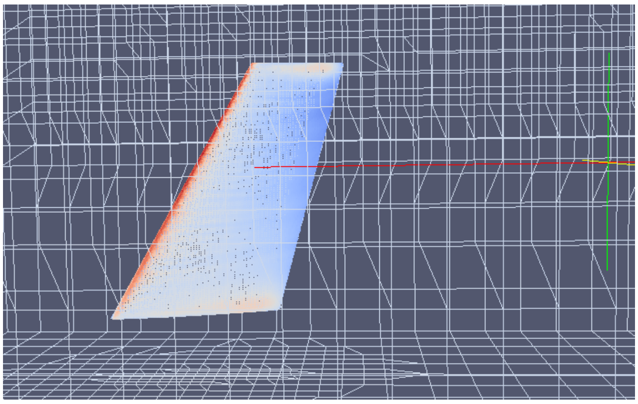


Figure 14: U variation 15 degree AOA (Top View)

Figure 15: U variation 15 degree AOA (Side View)

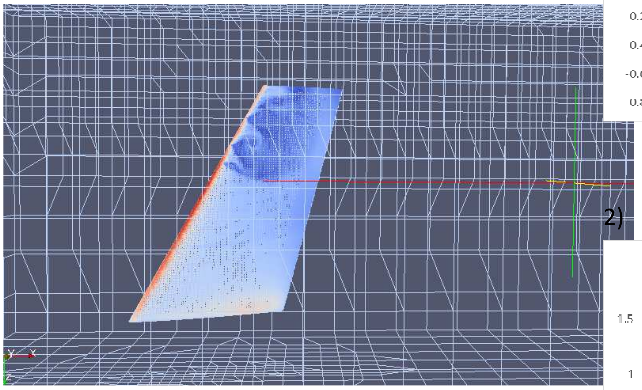


Figure 16: U variation 20 degree AOA (Top View)

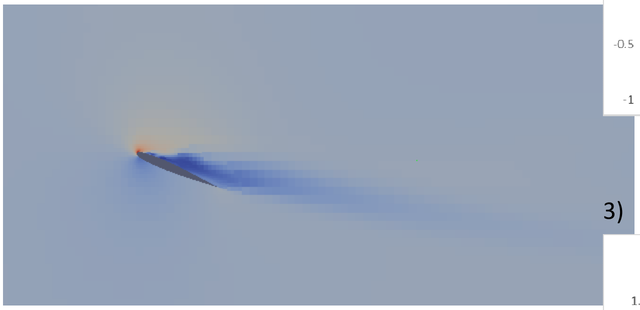


Figure 17: U variation 20 degree AOA (Side View)

As for visual comparisons, the selected span sectional locations were based on the available data from wind tunnel test results. Post simulation, the surface pressure data at those wingspan locations were sampled .which data are sampling and producing the output Cp values were done in ParaView. It was achieved by the usage" Plot over line" filters.

Following graphs are shown the pressure variations of both upper and lower surfaces between experimental and CFD results of all seven sections.

1) Section 1, y/C=0.2

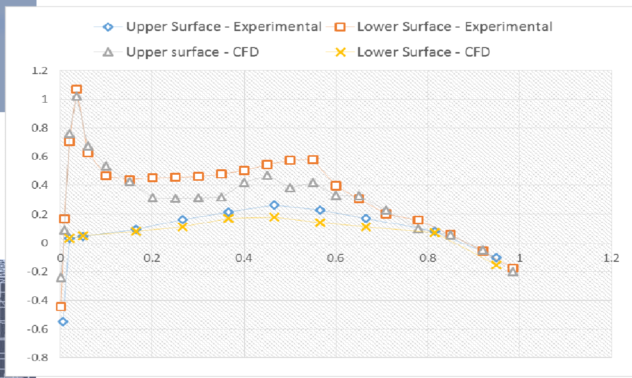


Figure 18: Data Comparison Section 1

2) Section 2, y/C=0.44

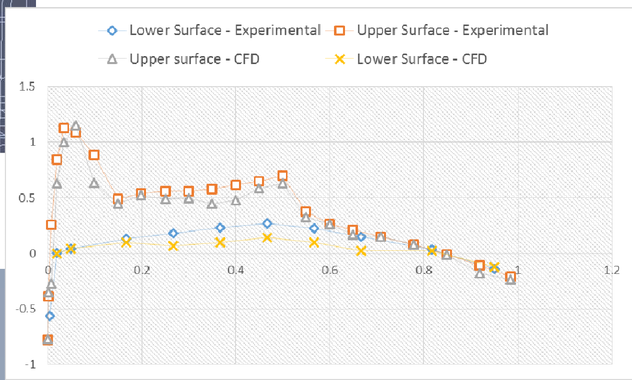


Figure 19: Data Comparison Section 2

3) Section 3, y/C=0.65

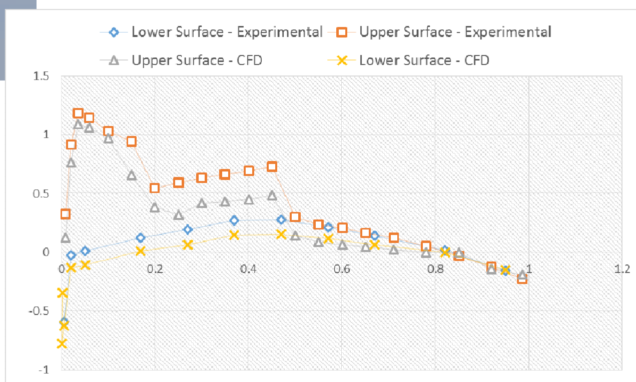


Figure 20: Data Comparison Section 3

4) Section 4, y/C=0.8

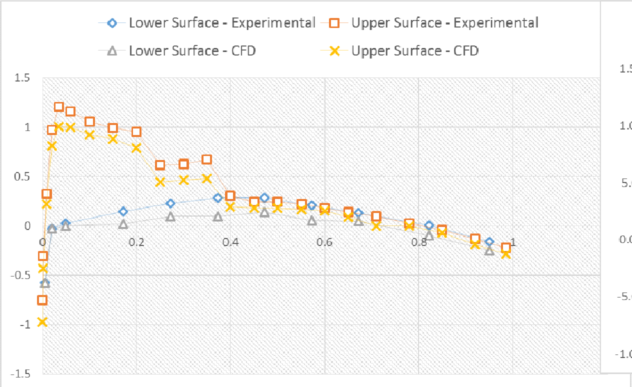


Figure 21: Data Comparison Section 4

5) Section 5, y/C=0.9

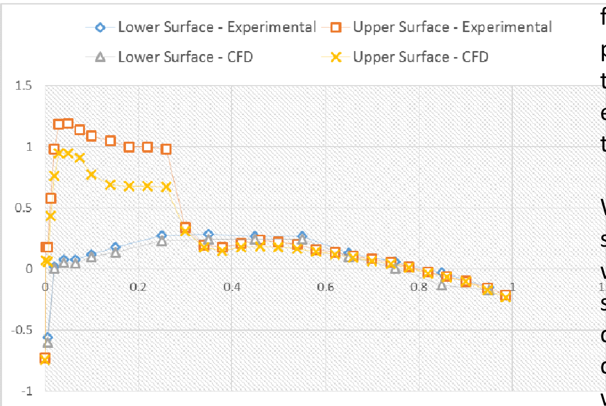


Figure 22: Data Comparison Section 5

6) Section 6, y/C=0.95

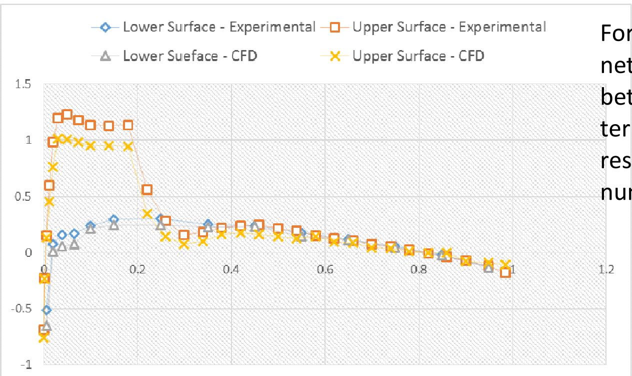


Figure 23: Data Comparison Section 6

7) Section 7, y/C=0.99

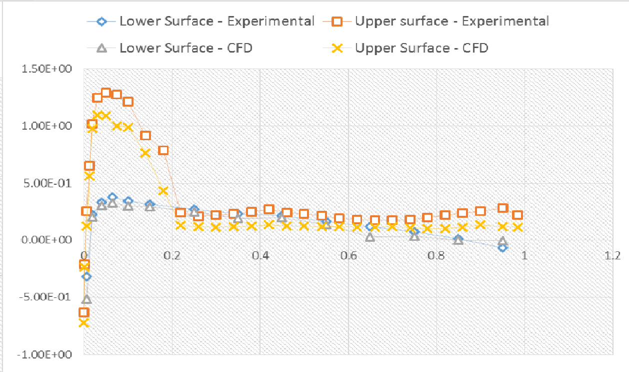


Figure 24: Data Comparison Section 7

For the more qualitative comparison; by taking the shapes. The average curve variation changing can be found to be less than 10%, 99% indicating a close proximity of CFD solutions as compared to wind tunnel data where at locations away from the extremities i.e. between stations 44%, 65% to 95%, the average differences are even lesser.

Where else in the CFD simulation, an ideal symmetrical boundary condition was applied on the wall at which the wing was attached. As for the station 99, the tip shaping during 3D modelling can differ slightly from the actual wing therefore a slightly different result expected. It is worth to mention as well, at 80% span, the second shock on the wing surface is not clearly visible when compared to wind tunnel test data. (RAhman, 2015)

Net Difference of area between the Cp values

For a more quantitative comparison; by taking the net different of the area formed under each curve between experiment Cp and numerical Cp values, in terms of percentages, below are the results. These results are calculated by using experiment Cp and numerical Cp values on to the “GRAPH” software.

Span Location	Top Different %	Bottom Different %	Average Different %
20%	16.21%	28.40%	22.31%
44%	15.45%	24.67%	20.06%
65%	32.23%	14.60%	23.41%
80%	25.23%	28.12%	26.67%
90%	25.29%	28.55%	27.07%
95%	25.67%	18.89%	22.28%
99%	33.17%	17.70%	25.43%

For the accurate results, the difference of the experiment Cp and numerical Cp values should be less. But here average different percentage is about 20% – 27%. It is not good. Reason for that high value is less number of iteration are conduct. If we could able to conduct 50000 of iteration, the difference between the experiment Cp and numerical Cp values may be minimum.

IV. CONCLUSION

This research project was evaluated the aerodynamics behavior of the OneraM6 wing by conducting Computational Fluid Dynamics (CFD) analysis while performing the CFD simulations in transonic region with K omegaSST turbulence models. The simulations for Onera M6 wing were quite successful and good agreement of the results with other studies and the experimental data was obtained. Turbulent flow over Onera M6 wing for angle of attack of 3,06° and Mach number of 0,8395, are presented. Completely unstructured mesh has been applied, with acceptably coarse elements in the vicinity of the wing. Nevertheless, the proposed method has confirmed its robustness, and ability to deliver the results of fair and satisfactory level of accuracy for the intended future applications, in the domain of much more complex fluid-structure interaction analyses of entire wing configurations. In this work, the compressible k-omegaSST turbulence model has been implemented in OpenFOAM for the first time.

In post processing this research is reviewed different angles of attack which are 3.06, 15 and 20 degrees and respective lift variations, drag variations, and pressure variations.it has been able to evaluate stalling requirements and shock interactions.

This project is developed around OneraM6 wing descriptions in transonic flow conditions.The forecasted values for aerodynamic efficacy and dimensionless parameters are lower than expected. It has been found this particular fact is directly related to computational limitations associated with CFD.

OpenFOAM usage may require a steep learning as the users need to be accustomed to text input files rather than the usual graphic interface which normally found on commercial CFD Open foam packages, this however, a very small price to pay when compared to its ability to expand without restrictions and its zero cost factor.

Acknowledgment

The authors gratefully acknowledge all the academic staff members of the Department of Aeronautical Engineering, General Sir John Kotelawala Defence University for the support given in this final research project.

REFERENCES

Gaultier, S., 2013. *ONERA-M6 Wing, Star of CFD.* [Online]
Available at: http://www.onera.fr/en/actualites/image-du-mois/onera-m6-wing-star-cfd

Gatski, C. L. R. a. T. B., 2014.. Open-source Analysis and Design Technology for Turbulent Flows. *AIAA paper,* p. 0243

v. Schmitt, F. C., 1979. *Pressure Distributions on the ONERA-M6-Wing at Transonic Mach Numbers,* s.l.: s.n.

Menter, F. R., 1994. Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. *AIAA ,* 32(8), p. 1605.

Rahman, U. A., 2015. *Validation of openFoam steady state compressible solver Rhosimplefoam,* Kuala Lampur: s.n.

Biography of authors

SH Subahan is a 4th year officer cadet undergraduate following the BSc Engineering degree in Aeronautical Engineering at General Sir John Kotelawala Defence University.

GDAL Gohumulla is a 4th year officer cadet undergraduate following the BSc Engineering degree in Aeronautical Engineering at General Sir John Kotelawala Defence University.

PDKL Sriyarathne is a 4th year officer cadet undergraduate following the BSc Engineering degree in Aeronautical Engineering at General Sir John Kotelawala Defence University.

WMSKB Wijayathunga is a 4th year officer cadet undergraduate following the BSc Engineering degree in Aeronautical Engineering at General Sir John Kotelawala Defence University.