Validations Of Openfoam® Steady State Compressible Solver Rhosimplefoam

Umran Abdul Rahman, and Faizal Mustapha

Abstract**—**OpenFOAM® steady state solver rhoSimpleFoam was tested. Reynolds Average Navier Stokes (RANS) coupled with kOmegaSST turbulent model simulation was performed. The simulation was based on 0.8395 Mach with Reynolds Number of 11.72 x 106 of air flow parameters over ONERA M6 wing section at 3.06 degrees angle of attack; an overall close fitting to the experimental values of Coefficient of Pressure (Cp) were observed. Some minor differences in a few pressure tapping ports, especially at 20% span cross section and at 99% span cross section were also noticed. The typically called "Lambda" shaped wave pattern was also observed on the wing top surface.

Index Terms**—**RhoSimpleFoam, Compressible, Transonic, Onera M6.

I. Introduction

OpenFOAM® is an open source CFD package which uses primarily Finite Volume Method (FVM) in solving CFD cases. Built in with its own meshing tools, multiple type of solvers and a third party data visualizer ParaVIEW®, these complete packages are there for the pickings of CFD users and researchers alike.

In this paper, the concentration will be to compare the results of wind tunnel testing against steady state compressible solver of OpenFOAM® i.e. rhoSimpleFoam. The test subject is the ONERA M6 wing panel where it was built and tested in a wind tunnel back in 1979 by Schmitt, V & F. Charpin [1]. This particular reference being selected as it has been the benchmark in validating many other CFD software [2]–[5] which provides solvers of similar capability. It has been known that the unpredictable nature of transonic flow poses a great challenge for any solver to closely predict the flow conditions and this is more so on a relatively complex boundary; therefore this validation test is an important step in establishing the solver for practical application in solving CFD cases of similar flow parameters.

II. Background

rhoSimpleFoam (from OpenFOAM® v2.3.0 package [6]) is the extension of incompressible steady state solver simpleFoam which uses the Semi-Implicit Method for Pressure Link Equations (SIMPLE) algorithm. This algorithm was introduced by Patankar & Spalding (1972) and later more detail utilization concept was presented by Joel H. Ferziger & Milovan PeriC [7]. The concept of solution is based on discretization of the flow equations (Navier Stokes equations in integral form – integral form as FVM is the base) into algebraic equations in the form of,

$$\mathbf{Ax} = \mathbf{b} \tag{1}$$

Where **A** is the coefficient matrix, **x** is the variable vector to be solved and **b** is the boundary conditions. Looking at the simplistic nature of (1), direct deduction can be made i.e. the number of discretized equations and unknowns must be equivalent to the number of finite control volumes in the calculation domain. Since the involved equations are non-linear to start with, direct solution will be too expensive therefore the sequential solution is adopted. The technique involved solving each variable one at a time while holding the rest of the dependent variables as constants. This is where there are two iteration loops occurred, the first loop is the inner iteration for obtaining the value of the individual variable, while the next is the outer iteration where the saved values from inner loop being used to solve the next variables.

Between each outer loop, if the solved variable values being used directly in solving the next variable, chances are it will cause the next variable values to be in an invalid physical domain. For example, the density variable may become a negative entity at certain region of the grid and this non-physical situation will definitely blow up the iteration process. To curb this, relaxation factors for each variable will be dialed in especially during the initial phase of the iteration.

Convergence of the solution can be observed thru the minimum residuals (the difference between previous variable values to current values) of each variable post inner loop and/or the consistency of the value of the variables itself thru the iterations. Later is, at times, is a better indication of convergence as there are occasions where individual variable residuals will not fall down to machine zero yet the value of the variables itself doesn't change any longer even though the iterations were being pro-longed.

III. Modelling

Apart from the given ONERA D top half airfoil coordinates (73 points), given also the plan form drawing which can be used to produce 3D drawings of the symmetrical wing. However, during production of the 3D model of the wing, there are a few discrepancies found on the given dimensions. The data presented in TABLE I shows the actual plan form dimensions as per the AGARD 138 report [1].

TABLE I
PLAN FORM DETAILS AS PER AGARD 138

Aspect Ratio	3.8
L.E sweep	30 degrees
T.E sweep	15.8 degrees

Manuscript sent December 6, 2014. This work was supported in part by the Ministry of Education, Malaysia under MyBrain15 scholarship for Industrial Doctorate Student (GS34199) University Putra Malaysia (UPM).

Umran is with the Malaysia Airlines. He is now the engineer in charge for A380 fleet while pursuing his doctorate studies with UPM (e-mail: umran.abdulrahman@malaysiaairlines.com).

Prof Dr. Faizal Musapha is with Department of Aerospace Engineering, UPM, Serdang, Selangor, Malaysia (e-mail: faizal@eng.upm.edu.my).

Taper Ratio	0.562
Mean Aerodynamic Chord (MAC)	0.64607 m
Semi span Length	1.1963 m
Semi span Area	0.7532 m^2
Root chord	0.8059 m
Tip chord (based on the above Taper Ratio)	0.452916 m
Rectangular length (dimension bound)	1.5787 m
Rectangular width (dimension bound)	2.7345 m
Tip shape - half round rotation of airfoil	N/A

By using the given taper ratio of 0.562, sweep angles and root chord, the semi-span area worked out to be 0.75296 m^2 and MAC to be 0.64590 m where these values are lower than the data presented in the report. The significant effect of these differences is the Reynolds Number calculation later for flow similarities and of course the net effect of the pressure gradient. In order to maintain the effective area and MAC as per the document, the taper ratio needed to be increased to 0.562516 instead of 0.562. Consequently the tip chord grows slightly to 0.453332 m instead of 0.452916 m resulting in MAC to be 0.646068 m and the effective area becomes 0.753209 m^2. By percentage of rounding off errors, this reduces the error down to only 0.0003%.

There are also issues in shaping the wing tip as there are no actual coordinates given, however the report do mention about the tip being formed by rotational extrusion of the tip airfoil half way. If this statement being applied directly, the sweep Leading Edge (LE) will meet with a forward bulge of the tip due to the rotation axis is in parallel with stream flow while the LE is slanted rearwards. From the report photo (Fig. 1), this doesn't seem to be the case. To alleviate this issue, the rotation axis maintained, however, as the tip airfoil being rotated; at each of the rotated tip airfoil coordinate, rearwards transformation being applied according to each coordinate swept angle from the root airfoil coordinates, resulting in a smooth LE of the tip against the swept angle, this is however purely an estimation. Fig. 2 shows the tip design while Fig. 3 shows the plan form dimension of the generated 3D model.

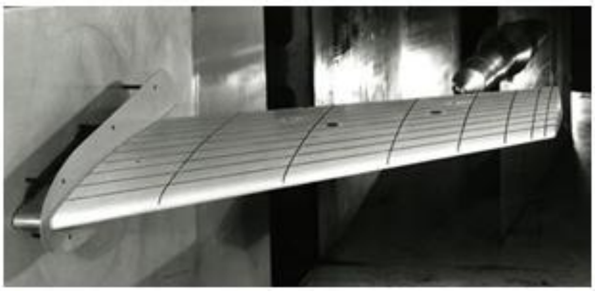

Fig. 1 Actual Onera M6 Wing, Note The Tip Shape

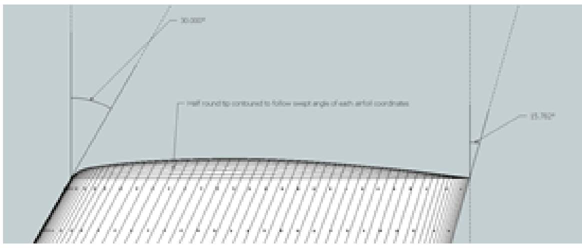

Fig. 2 Tip Design

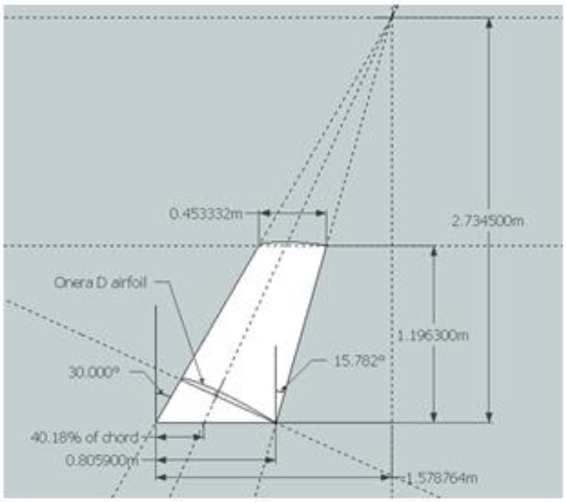

Fig. 3 Plan Form

IV. MESHING

From the given Onera D airfoil coordinates; by extrusion, swept translation, tip scaling and lastly rotation, the basic 3D volume coordinates were created based on Fig. 3 plan form dimensions. These initial volume coordinates together with the extrusion path coordinates up to the domain boundary faces being fed into the block meshing tool (blockMesh) of OpenFOAM® in creating a fully structured mesh. Special care was given in ensuring:

1. All cells grow in volume towards external boundaries with controlled aspect ratio.
2. Proper ratio of cell height and width between adjacent cells.
3. Flatness of cell walls.
4. Skew angle of cell walls being kept minimal as the cell grows towards external boundaries.
5. Only hexahedral cells used.

Mesh Statistic

Points: 690279; Faces: 2032484; Internal Faces: 1994476; Cells: 671160.

Overall Domain Dimension: 30 m x 30 m x 14 m (length x height x width).

Wing Location: Wing root planted on symmetry boundary where the root LE is at 10m from inlet and 15m equidistant from top and bottom boundaries, refers Fig. 4.

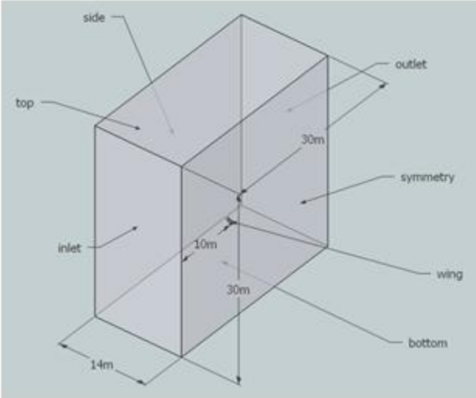

Fig. 4 Overall Domain

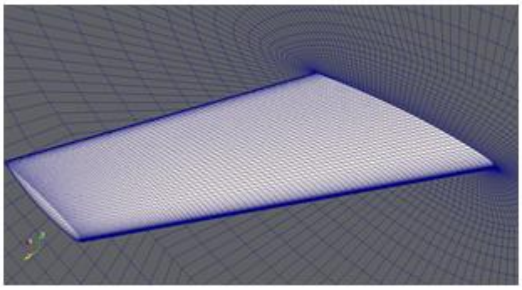

Fig. 5 Surface Mesh

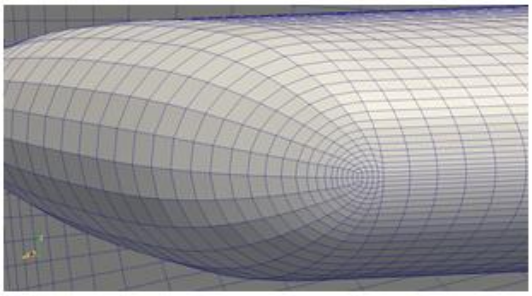

Fig. 6 Nose Tip

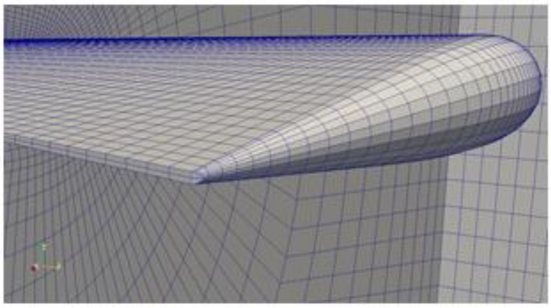

Fig. 7 Tail Tip

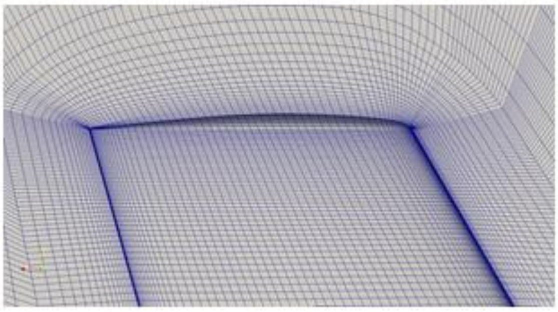

Fig. 8 Horizontal Cut (Cell Growth)

V. Simulation Parameters

Mach number: 0.8395; Angle of Attack: 3.06 degrees; Reynolds Number: 11.72 x 10^6. These parameters were given in the report where the stagnation temperature ranges between 292 and 315 Kelvin (K). In order to match the flow parameters, Reynolds Number equation,

$$Re = \frac{\rho VL}{\mu} \tag{2}$$

Where V is velocity, L is the effective length (MAC in this case, which is 0.646068 m), ρ is density and μ is dynamic viscosity, coupled with Mach number equations,

$$M = \frac{V}{a} \tag{3}$$

Equation of State,

$$\rho = \frac{p}{RT} \tag{4}$$

Sonic equation,

$$a = \sqrt{\gamma RT} \tag{5}$$

Where a is the local sonic velocity, R is the gas constant (287 J/Kg.K), γ is the ratio of thermal constants (1.4) and T is the static temperature, and also the Sunderland's equation [8] which linked the temperature to dynamic viscosity,

$$\mu = \frac{CT^{3/2}}{T+S} \tag{6}$$

Where C is the constant (1.458 x 10^{-6} kg/ms√K), S is the Sunderland's Temperature (110.4 K) and by combining the equations from (3) to (6) into (2), fixing a pressure value (taken as standard sea level air pressure of 101325 N/m^2) and inserting the rest of the known variable values, the Reynolds Number equation reduced to the function of temperature only.

By applying iterative procedure, a temperature of 305.7335559 K was found to satisfy the effective flow Reynolds Number. By substituting this temperature value into (5) and later into (3) and by using the given Mach number, the relative air velocity was determined to be 294.236957 m/s. These 3 values (temperature, pressure and velocity) were used as the boundary conditions of the calculation domain where the velocity being vectored accordingly to achieve 3.06 degrees angle of attack.

A minor argument can be made here; there is no mentioned of actual static air pressure of the flow at Mach 0.8395 but it does mention about dynamic air pressure between 1600 to 64000 N/m^2 for the whole tested range of Mach numbers between 0.27 to 1.33. It does also mention the stagnation pressure values fall within 30,000 to 210,000 N/m^2 for Mach 0.8. With this info alone, it is relatively vague in determining the actual static air pressure at the inlet boundary, therefore the question will be, how exactly we know that by taking the standard sea level ambient air pressure is sufficed to mimic the inlet static air pressure during the wind tunnel testing?

First of all, we don't, however, instead of simulating air being blown over the wing such as what the tunnel testing did; this simulation can be seen as moving the wing itself into a static air where the actual velocity of the wing and the resultant airflow temperature which passes the wing can be controlled at the domain boundary level. This is where the value hold and the effective Mach and Reynolds Number matched.

VI. Turbulence Modelling

Noting that the flow is in a relatively high Reynolds Number region, turbulent flow is expected. As this simulation is a RANS type simulation, turbulent eddies can't be fully resolved as in Direct Numerical Simulation (DNS), therefore it requires modeling. The trade off here is, due to some level of approximations are needed in turbulent modeling, expectedly, some level of precision will have to be sacrificed. This sacrificial however, resulting in the grid count that doesn't need to be at a level where a typical modern desktop will not be able to cope with in order to obtain a reasonable flow prediction. In this case study, the turbulence was modeled using k-omega SST (Shear Stress Transport) developed by Menter, F.R [9] where the inlet boundary values of k and omega were generated using empirical estimations based on [10].

VII. Results & Discussions

The simulation was allowed to run up to 50,000 iterations as this will permit the observation on residual trending together with the consistency of the solved variables.

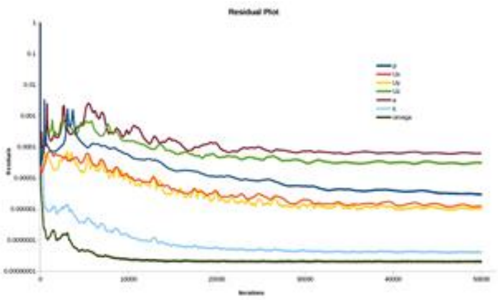

Fig. 9 Residual Trending

Based on Fig. 9, it can be observed that the simulation heading into a converged solution where the rippling frequency of each variable residual continually trending towards steady state as the iterations progressed. With reference to Fig. 10 and 11, it can be readily seen that the Cp (inversed) at the selected cross sections of the wing marked very subtle differences between 30,000 and 50,000 iterations. This is another indication depicting a converged solution. From the same figures as well, the sharply defined 'λ' (lambda) shaped Cp gradient on the top wing can be observed, indicating the expected occurrence of shock waves on the top surface were fully resolved.

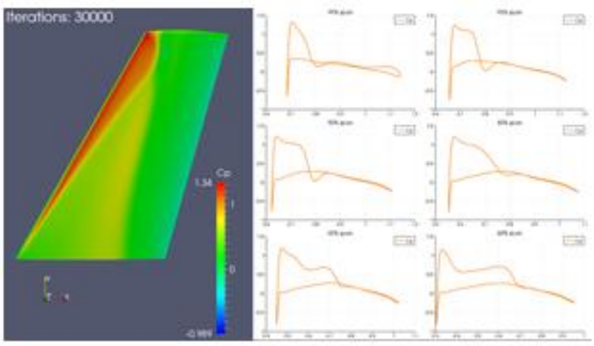

Fig. 10 - Results at 30,000 Iterations

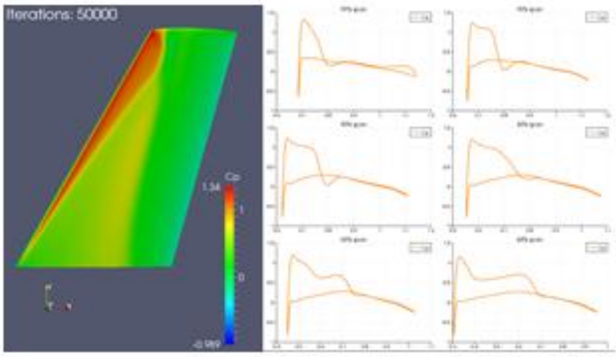

Fig. 11 Results At 50,000 Iterations

As for numerical 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 and the inversed Cp values were generated by passing the data through (7).

$$-Cp = \frac{2}{\gamma M_{\infty}^{2}}\left(1 - \frac{p}{p_{\infty}}\right) \tag{7}$$

These data sampling and producing the output Cp values were done in ParaView®. It was achieved by the usage of 'Calculator' and 'Plot On Intersection Curves' filters. As the available wind tunnel data at each selected location on the span were normalized in terms of its chord, these numerically obtained data were normalized as well in order to have a consistent plot axis. Both data (at the same location) were plotted against each other using Microsoft Excel and can be inspected in Fig. 12 through Fig. 18.

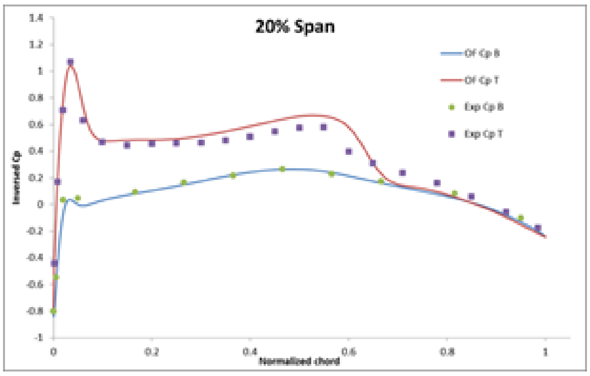

Fig. 12

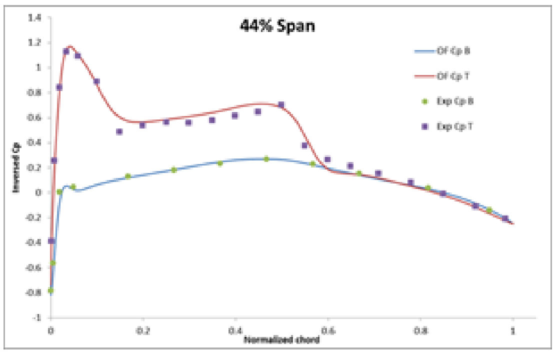

Fig. 13

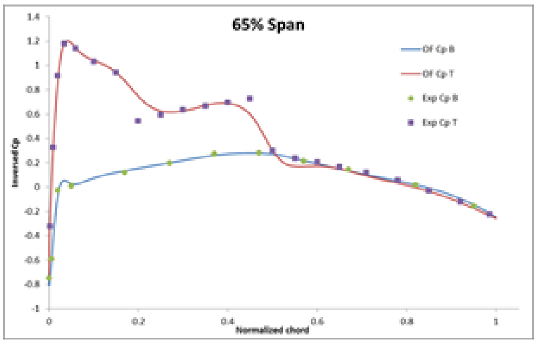

Fig. 14

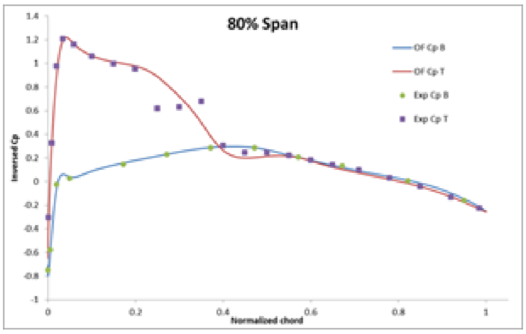

Fig. 15

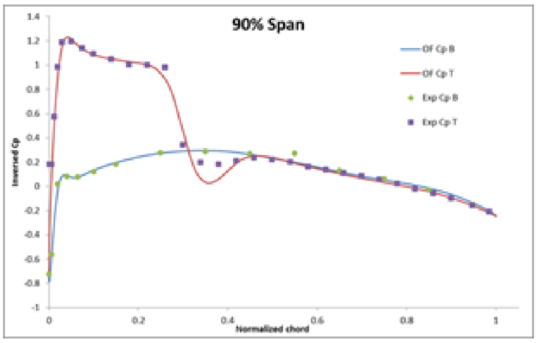

Fig. 16

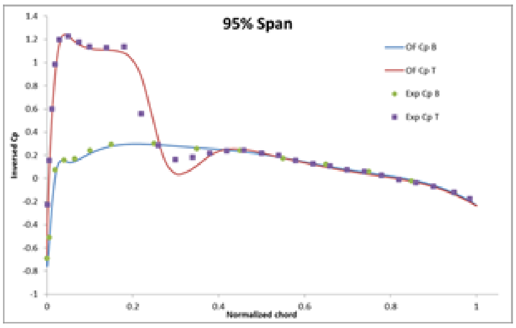

Fig. 17

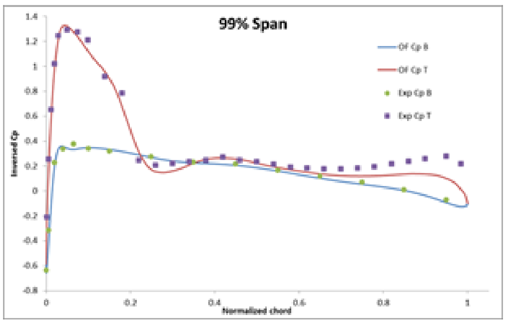

Fig. 18

Based on the comparative presented data, the numerically obtained solutions by rhoSimpleFoam solver coupled with k-omegaSST turbulent model has the ability to closely predict the resultant surface pressure from a transonic flow condition. For a more quantitative comparison; by taking the nett different of the area formed under each curve between experiment Cp and numerical Cp values, in terms of percentages, TABLE II below was produced.

TABLE II
NETT DIFFERENCE OF AREA BETWEEN THE CP CURVES

Span Location	Top Diff (%)	Bottom Diff (%)	Average Diff (%)
20%	6.717	9.054	7.886
44%	2.570	1.462	2.016
65%	0.775	5.314	3.044
80%	0.570	5.799	3.185
90%	3.168	2.066	2.617
95%	0.715	1.949	1.332
99%	11.814	3.862	7.838

The average differences can be found to be less than 10% 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 to 95, the average differences are even lesser.

By going back to Fig. 12 and Fig. 18, minor discrepancies can be seen and this can be confirmed by making cross reference to TABLE II. At these extremities locations, those differences may be attributed to the differences in modeling. In Fig. 1, the actual tunnel testing of the semi span wing, one can see a divider plate was installed at the root of the wing. The dimension and thickness of this plate is not available and so does its distance from the tunnel wall. On this plate, there will be a buildup of boundary layer which grows in the span wise direction. This boundary layer may not only affecting the pressure gradient on the wing surface, but it may also (to a certain extend) affecting the location at which the formation of the shock waves on the wing surface. 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 (Fig. 18), the tip shaping during 3D modeling can differ slightly from the actual wing therefore a slightly different result expected.

It is worth to mention as well, at 80% span (Fig. 15), the second shock on the wing surface is not clearly visible when compared to wind tunnel test data. The contributing factor to this phenomena is the junction of λ wave (the point where all the three arms met) on the wing top surface (refer Fig. 10 & 11) was shifted more towards the wing root in CFD simulation as compared to wind tunnel testing. This shift can be the product from inaccuracy of tip shaping as well as the approximations involving turbulence modeling.

VIII. CONCLUSIONS

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 packages; this however, a very small price to pay when compared to its ability to expand without restrictions and its zero cost factor.

In engineering, the results obtained from this validation work are more than enough in meeting the industry standards, however, from scientific point of view, perhaps more can be pursued such as running a DNS simulation on the 3D mesh generated from laser scanning of the actual ONERA M6 wing panel.

For all practical purposes, this presentation has established rhoSimpleFoam, the steady state transonic solver from OpenFOAM® is on par with (if not better than) many other commercial codes of similar capability.

ACKNOWLEDGMENT

The Author would like to thank the Board Members of School of Graduate Studies, University Putra Malaysia (UPM). Thanks also to D.Eng program advisors Dr. Mohamed Thariq and Dr. Azmin Shakrine for the guidances given in preparing this paper. To Prof. Dr. Abdul Aziz and Prof. Dr. Nik Abdullah from University Malaysia Pahang (UMP), thanks for the interest in Author's work. Special thanks to The Ministry of Education, Malaysia for the granted scholarship under MyBrain15 program.

REFERENCES

[1] Schmitt, V. & F. Charpin, "Pressure Distributions on the ONERA-M6-Wing at Transonic Mach Numbers," Experimental Data Base for Computer Program Assessment. Report of the Fluid Dynamics Panel Working Group 04, AGARD AR 138, May 1979.

[2] Christopher L. Rumsey and Thomas B. Gatski, "Summary of EASM Turbulence Models in CFL3D With Validation Test Cases", NASA/TM-2003-212431, June, 2003.

[3] Tatum, K.E. & J.W. Slater, "The Validation Archive of the NPARC Alliance", AIAA Paper 99-0747, January, 1999.

[4] Taku Nagata et al, Validation of new CFD Tool Using Non-orthogonal Octree with Boundary-fitted Layer Unstructured Grid – AIAA 2012-1259, 2012.

[5] Francisco Palacios et al," Stanford University Unstructured (SU2): Open-source Analysis and Design Technology for Turbulent Flows", AIAA paper 2014-0243, January 2014.

[6] http://www.OpenFOAM.org/download/ubuntu.php (25/07/2014)

[7] Joel H. Ferziger and Milovan PeriC, Computational Methods for Fluid Dynamics, ISBN 3-540-42074-6, 3rd Edition, 2002.

[8] The viscosity of gases and molecular force. Philosophical Magazine, S.5, volume 36 (1893), 507-531. (Obtained the ref summary from http://www.cfd-online.com/Wiki/Sutherland's_law - 22/04/2014)

[9] Menter, F. R., "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," AIAA Journal, Vol. 32, No. 8, August 1994, pp. 1598-1605.

[10] http://www.cfd-online.com/Wiki/Turbulence_free-stream_boundary_conditions - 18/07/2013.

The Author was born in 1967, Apr12. Received Aircraft Maintenance Engineer's License in 1992 from Civil Aviation Authority, United Kingdom. Obtained a Bachelor in Mechanical Engineering from University Kebangsaan Malaysia (1999) and later completed a Masters of Science course from Institute of Technology Bandung, Indonesia in year 2006. Currently pursuing his Doctorate in Industrial Engineering from University Putra Malaysia.

Work as an Engineer for Malaysia Airlines since 1992 and currently being tasked to manage the A380 fleet. Have produced unpublished reports on the fleet performance and defect analysis for aircraft manufacturer review, which include identifying root causes of certain critical defects on the fleet.

Mr. Umran Abdul Rahman being engaged in consulting work on UAVs by UPM, open source CFD system implementation in UMP and also being engaged by an independent contractor to perform aerodynamic analysis on UAV platform suggested for Ministry of Defense, Malaysia.

Prof. Dr. Faizal Mustapha is a distinguish Professor in UPM on Structural Analysis. Obtained his PhD in Structure Health Monitoring from University Of Sheffield, U.K. Currently engganging in various sectors which needed consultation on structure condition monitoring.