

# Verification and Validation of Flow Over a 3D ONERA Wing using CFD Approach

Muhammad Umer Sohail<sup>1</sup>, Asad Islam<sup>2</sup>

**Abstract**—ONERA M-6 wing is definitive CFD validation case for peripheral flows. Due to the reason of its unpretentious geometry associated with intricacies of transonic flow (i.e. local supersonic flow, shocks and turbulent boundary layers' separation). Transonic flow over the swept wing ONERA M-6 has been studied. The location of shock waves and the supersonic region on the wing is computationally analyzed. Over the years it has practically developed a standard for CFD codes since of its admittance as a validation case in frequent CFD research papers. A 3D flow simulation was accomplished in Fluent® on ONERA M6 wing by using S-A turbulence model. Flow was modelled as being transonic and compressible at an AOA of 3.06 degrees with Reynolds number of 11.72e+6 having Mach of 0.8395. The CFD results thus obtained will be validated against the experimental data that are available for 1/5th span wise location of the wing

**Index Terms**—3-D, Fluent®, ONERA, S-A, validation, wing

## I. INTRODUCTION

Since 1930s most widely studied and investigated in the fields of flight is transonic regime. Due to the formation of shock waves, turbulence boundary layers interaction and other factors that affects the transonic flight which leads to flow separation and causes instabilities on large scale. Different experiments and computational analysis has been performed to investigate the transonic flow over the ONERA wing. Boeing 777, 747 etc. are the subsonic modern airplane that is being cruise at 0.85 Mach number. The accurate calculation for the airfoil and wing characteristics in recent times is CFD analysis. Computational Fluid Dynamic (CFD) is latest and modern technique for solving the real life problems regarding fluid dynamics (both compressible and incompressible fluids) through numerical simulation. ONERA Aerodynamics department designed an ONERA M6 wing in 1972. As it was being designed on experimental geometry for reviewing high Reynolds number and three dimensional flows with some multifaceted, complex flow circumstances [1], comparatively unpretentious geometry, complex flow physics, and

accessibility of experimental data. At a transonic Mach number, in inviscid flows this trial circumstance is being executed [2]

## II. FLOW DESCRIPTION

Presently, the simulation of Computational Fluid Dynamics uses the flow field phenomenon of Test 2308 as shown in Table 1. At high Reynolds number of 11.72 million which is based on the mean aerodynamic chord of 0.64607 meters the table shows the details of flow circumstances [3]

Table 1  
Flow conditions for Test 2308.

<i>Angle-of-Attack (AOA) (deg)</i>	<i>Angle-of-Sideslip (deg)</i>	<i>Mach Number.</i>	<i>Reynolds Number</i>
3.06	0.0	0.8395	11.72E+06

The ONERA M-6 wing is a swept, semi-span wing with no convolution [4]. Its specifications are shown below:

Table 2  
ONERA Wing Specification.

<b>Leading edge sweep</b>	30 degrees
<b>Trailing edge sweep</b>	15.8 degrees
<b>Span, b</b>	1.1963 meters
<b>Mean aerodynamic chord, c</b>	0.64607 meters
<b>Aspect ratio</b>	3.8
<b>Taper ratio</b>	0.562

The CFD results thus obtained will be validated against the experimental data that are available for 1/5th span wise location of the wing. Furthermore, the CFD results can also be compared with the NASA's WIND ® simulation results.

<sup>1</sup>Muhammad Umer Sohail is a Mechanical Engineer and currently pursuing PhD degree program in Aerospace Engineering from IST, Pakistan. phone:+92-344-5266876; email: engr.mu.sohail@gmail.com

<sup>2</sup>Asad Islam has master's degree in Aerospace Engineering from IST, Pakistan, E-mail: engr.asad.islam@live.com

### III. COMPUTATIONAL SETUP

#### A. Geometry

ANSYS Design Modeler® is a user-friendly CAD modelling software for creation of geometries. Figure 1 shows the 3D geometry of ONERA M6 wing, which was created using ANSYS Design Modeler

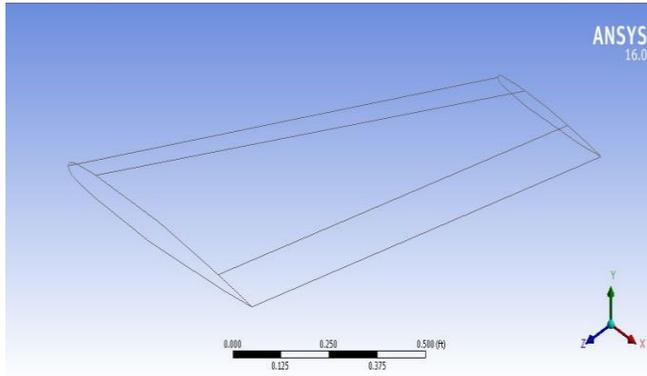


Figure 1: ONERA M6 Wing CAD Geometry

Figure 2 shows the enclosure for the wing depicting the inlet, outlet, and the far and symmetry sides.

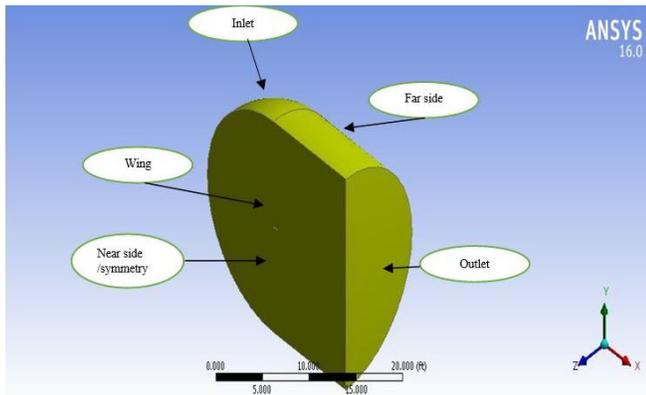


Figure 2: Enclosure of Wing for computational setup

#### B. Mesh

Mesh is of considerable importance for computational purposes. The meshing is done so as to cover whole domain completely and that there is no free space between the cells. Furthermore, cells do not overlap each other and there are no negative volumes present in the domain. A good quality mesh can give better results, with good comparison with the available experimental or CFD results; for validation and verification respectively. Out of Structure, Un-structural and Hybrid meshing, the structured meshing is preferred because the structured discretization system discretize the boundary surface of flow domain consuming quadrilateral which is named as the surface grid and captivate the whole geometry with hexahedral [5] [6]. Structured meshes are visualized by

steady connectivity. The conceivable section adoptions are quadrilateral in two dimensional and hexahedral in three dimensional. This is extremely space effective, i-e. Subsequently the region associations are well-defined by storage procedure [7]

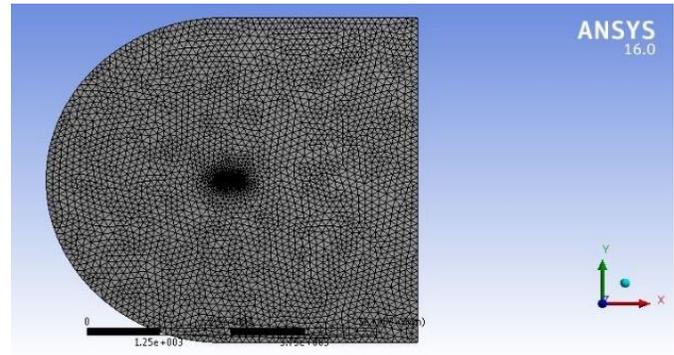


Figure 3: Mesh of wing's enclosure

Mesh details are displayed in table 3.

Table 3  
Mesh Specification

Type	Number of Nodes	Number of Elements
Mesh	208486	941463

Figure 3 and 4 shows the mesh and its refinement near the edges of the wing.

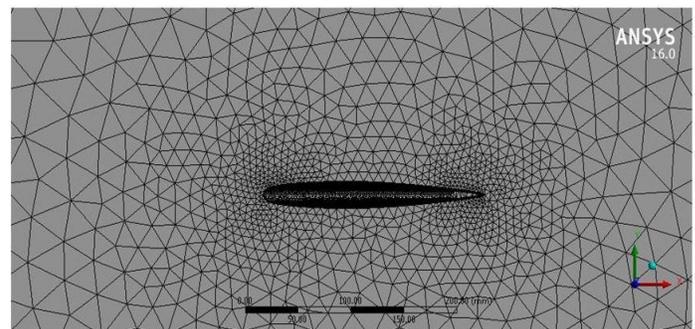


Figure 4: Mesh refinement

#### C. Boundary Conditions:

The next task to proceed solving is to provide inlet and outlet boundary conditions. These conditions provide the solver the inlet condition to flow (inlet pressure). We can also provide inlet velocity, inlet mass flow rate or Cartesian velocity component or cylindrical velocity component. Also outlet conditions are provided at outlet boundary where we can define exit static pressure or exit mass flow rate as shown in Table 4 [8]

Table 4  
Boundary Conditions

Boundary	Type	Conditions
<b>Inlet</b>	Pressure far field	Flow direction along x-component: 0.9986
<b>Outlet</b>	field	Flow direction along y-component: 0.0534
<b>Far side</b>		Temperature (R) = 460 K Gauge Pressure (psi) : 45.829 Mach number: 0.8395
<b>Near side</b>	Symmetry	
<b>Wing surface</b>	Wall	
<b>Wing tip</b>	Wall	

Whereas the fluid properties are as under:

Table 5  
Fluid Properties

Fluid	Properties
<b>Air</b>	Density: (as of Ideal gas) Viscosity: 1.09329e-05 lbm/ft.s

*D. Solver & Turbulence Model:*

Pressure based solver was used for solution with a Pseudo-Transient approach having Spalart Allmaras (1eqn) as turbulence model. This S-A turbulence mode has been specially developed for the aerospace applications that are wall bounded and are subject to adverse pressure gradients. [9]

IV. VERIFICATION & VALIDATION OF CFD RESULTS

A. Verification

NASA CFD results from WIND® software are available for verification of Cl and Cd CFD results. So, Table 6 shows a comparison of our Fluent CFD results with the NASA CFD software results [10]

Table 6  
CFD comparison of NASA and Author's results

	Cl	Cd	%error Cl	%error Cd
<b>NASA CFD</b>	0.1410	0.0088		
<b>Fluent (CFD)</b>	0.12708	0.01085	<b>9.869 %</b>	<b>23.319 %</b>
	4	2		

The Figures 5 to 7 revealed the pressure coefficient that contours on the wing. There is a clear formation of shock in Figure 6 which also shows the analytical comparison of Cp contours at the symmetry plane of the wing with NASA's mesh CFD [11].

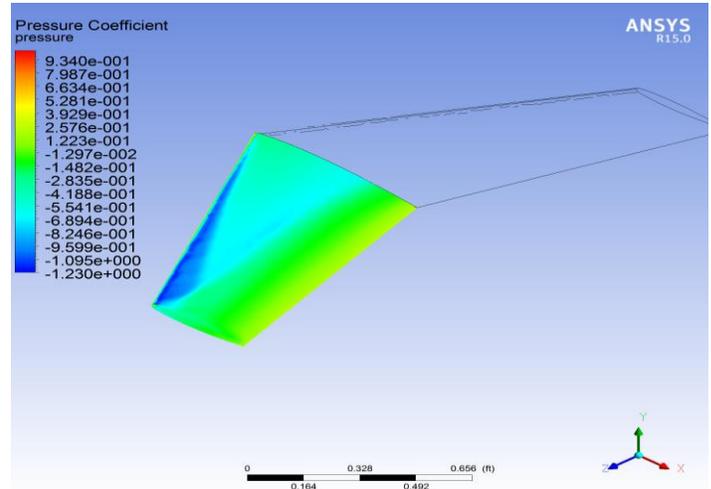


Figure 5: Pressure coefficient contours on the wing

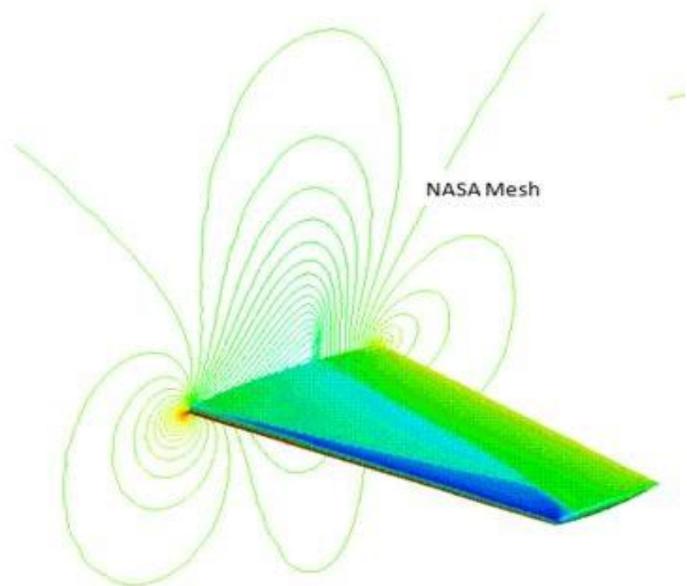


Figure 6 (a): NASA Pressure Distribution

B. Validation of CFD Results

For Validating CFD results with the available experimental data; a polyline was constructed at span wise location of 0.2 ft, to compare Cp (Pressure coefficient) results at this location on the wing. Figure 10 shows that CFD results of Cp are in upright arrangement by the available experimental facts and data at this location that validated the results [13]

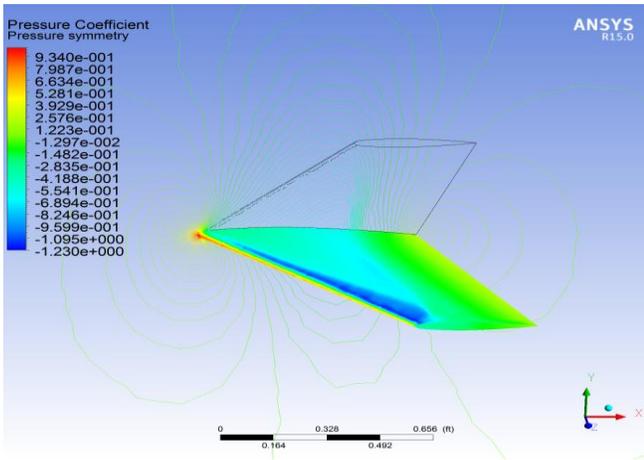


Figure 6 (b): Analytical comparison with NASA CFD results

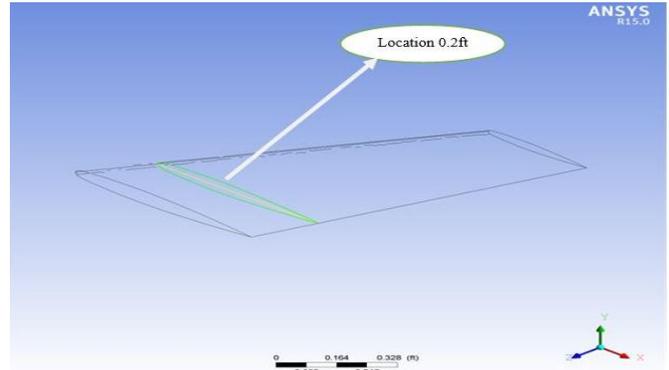


Figure 9: Polyline at z=0.2 ft for Cp results validation

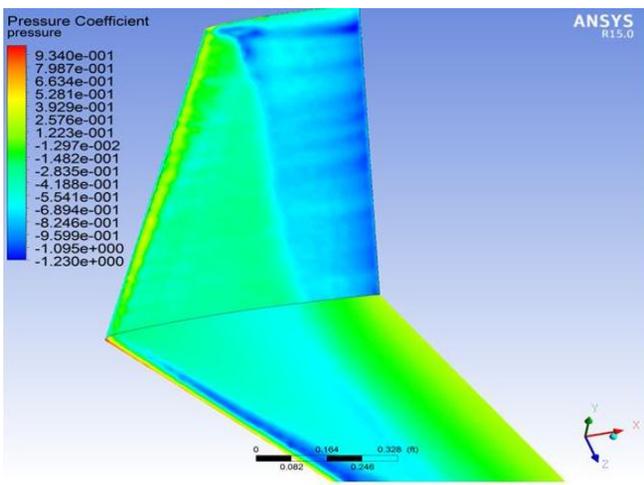


Figure 7: Cp contour with shock formation

Figure 7 depicts the shock formation on the surface of the wing. Furthermore, to visualize the effect of boundary layer; Ma # contours at the symmetry plane were developed which is shown in Figure 8. It shows that there is a thin boundary layer in the highlighted region besides the boundary layer thickens later the shock [12]

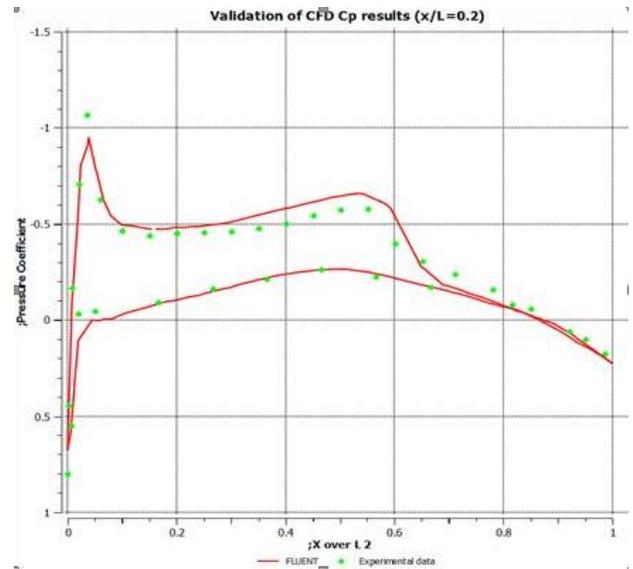


Figure 10: Validation of Cp (at 1/5th span wise location) CFD results by experimental data

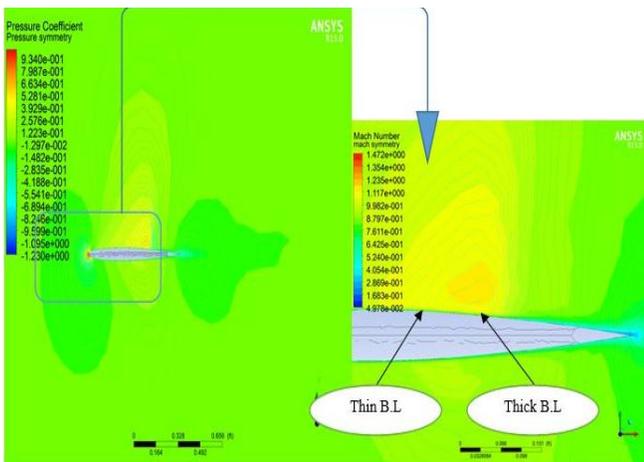


Figure 8: Boundary Layer Development

## V. CONCLUSIONS

The research shows reasonable positive computational results of Transonic ONERA wing. The result indicates good agreement with the experimental data. The decent validation of CFD result of  $C_p$  distribution with available experimental data is quite encouraging. A 3D flow simulation was executed on ONERA M6 wing through using S-A turbulence model. Flow was modelled as being transonic and compressible at an AOA of 3.06 degrees with Reynolds number of  $11.72e+6$  having Mach of 0.8395 [14]

1. Results show that the wing experiences supersonic conditions, a shock and a boundary layer separation as depicted in figures 7 & 8.
2. The CFD results were validated and it showed excellent arrangement with the available experimental data as depicted in Figure 10 [15].

## VI. REFERENCES

- [1] D. McBride, "A coupled finite volume method for the solution of flow processes on complex geometries," *International Journal for Numerical Methods*, 2007.
- [2] [Online]. Available: [www.efluid.com](http://www.efluid.com).
- [3] "Wing at Transonic Speed," [Online]. Available: [www.dept.aoe.vt.edu/~mason/Mason\\_f/ConfigAeroTransonics.pd](http://www.dept.aoe.vt.edu/~mason/Mason_f/ConfigAeroTransonics.pd).
- [4] [Online]. Available: [www.grc.nasa.gov](http://www.grc.nasa.gov).
- [5] J. Blazek, "Principal of Grid Generation," in *Computational Fluid Dynamics Principles and Application*, vol. 1, Elsevier, 2001.
- [6] Lambropoulos, "Acceleration of a Navier Stokes Equation Solver for Unstructured Grods using Agglomeration Multigrid and Parallel Processing," *Applied Mechanics and Engineering*, 2013.
- [7] Chitale, Kedar, Onkar Sahni, Saurabh tendulkar, Rocco, Shephard, "Boundary Layer Adoptivity for Transonic Turbulent Flows," *AIAA Computational Fluid Dynamics Conference*, vol. 21, 2013.
- [8] J. Dandois, P. Molton, A.Lepage, A. Geeraert, "Buffet Characterization and Control for Turbulent Wing," 2013.
- [9] in *Submitted to Rochester Institute of Technology*.
- [10] Jonsson, "Aerodynamic Optimization of Wings by Space Mapping," *AIAA Aerospace Science*, 2013.
- [11] Durrani, Naveed, Ning Qin, "Comparison of RANS, DES and DDES Results for ONERA M6 Wing at Transonic Flow," *AIAA Aerospace Science Meeting Including the New Horizon*, vol. 49, 2011 .
- [12] T.Economon, 2014. [Online]. Available: [www.adl\\_public.stanford.edu/docs/display/susquared/tutorial+2+-Inviscis+ONERA+M6](http://www.adl_public.stanford.edu/docs/display/susquared/tutorial+2+-Inviscis+ONERA+M6).
- [13] Lo, S.C, "Development of an Unstructured Hinite Volume flow solver for Aerodyanamic Applications," *AIAA Aerospace Science Meating Including the New Horizon*, no. 51, 2013.
- [14] Palacios, Francisco, Karthik Duraisamy, Aniket Aranake, Sean R.Copelant, Thomas D.Economon, "Analysis and Design Technology for Turbulent Flows," *Aerospace Science Meeting*, no. 52, 2014.
- [15] Zhang, Laiping, Zhong Zhao, Xinghua Chang, Xin, "A 3D Hybrid Grid Generation Technoque and A Multigrid/Parallel Algorithem," *Chinese Journal of Aeronautics*, 2013.