

NUMERICAL 3D TRANSONIC FLOW SIMULATION OVER A WING

Cvetelina VELKOVA

Department of Technical Mechanics, Naval Academy Nikola Vaptsarov, Varna, Bulgaria
(cvetelina.velkova1985@gmail.com)

DOI: 10.19062/1842-9238.2017.15.3.1

Abstract: *This scientific paper presents performed numerical 3D transonic flow simulation over a wing. It is developed a suitable numerical model of the wing and it is created a three-dimensional mesh around the wing using the available techniques in ANSYS software. Also, it is obtained iterative convergence by using recommended solver settings. The performed numerical 3D transonic flow simulation over a wing give the opportunity to visualize 3D flow characteristics to gain physical insights. The main goal is to determine whether the numerical flow simulations over the wing performed by computational tools provide appropriate approaches for calculations of the complex 3D transonic flow characteristics. The main value of the paper is that the obtained results with the realized numerical flow simulation are compared with the experimental data and NASA CFD results using the source [1].*

Keywords: *aerodynamics, wing, ANSYS, lift and drag coefficients, transonic flow*

1. INTRODUCTION

Using the approach developed by Cornell University, [1] it is performed a 3D transonic turbulent CFD Simulation using ANSYS 15 and the solver is Fluent. For the properly investigation of the flow around the wing it has been created a three-dimensional mesh using the available techniques in ANSYS software. The performed numerical 3D transonic flow simulation over a wing give the opportunity to visualize 3D flow characteristics to gain physical insights. The main goal is to determine whether the numerical flow simulations over the wing performed by computational tools provide appropriate approaches for calculations of the complex 3D transonic flow characteristics. The main value of the paper is that the obtained results with the realized numerical flow simulation are compared with the experimental data and NASA CFD results using the source [1].

2. PROBLEM SPECIFICATION

Using the approach in source [1] the performed numerical 3D transonic flow simulation over the wing is trying to obtained the results obtained by NASA using the wind, and this is verified by comparison of the numerical results with the experimental data, [2].

Cited source [5] the Onera M6 wing is a classic CFD validation case for external flows because of its simple geometry combined with complexities of transonic flow (i.e. local supersonic flow, shocks, and turbulent boundary layers separation).

Flow over the Onera M6 wing is transonic and compressible. Quoting [6] conventional CFD methods are applicable in this case because they required a calculation for the entire three-dimensional field about the body.

The wing flow experiences supersonic conditions, a shock and boundary layer separation. The wing has no twist. There is no side-slip in the simulation. The flow conditions are given below at Table 1:

Table 1 Flow Conditions, [1]

Mach	Reynolds Number	Angle of Attack (degrees)	Angle of Side-slip (degrees)
.8395	11.72E6	3.06	0

2.1. Mathematical Model

The performed numerical simulation is governed by the continuity, Navier-Stokes (momentum conservation) and energy equations. But the flow is turbulent, that's why it was used Reynolds Average version of their equations. Therefore it was used Spalart-Allmaras turbulence model. In the beginning, it has six variables to solve for: 3 components of velocity, pressure, temperature, and kinematic eddy viscosity. The equations are:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0 \quad (1)$$

$$\rho \left(\frac{\partial U_i}{\partial t} + \frac{\partial}{\partial x_j} (U_i U_j) \right) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left(2\mu S_{ij} - \overline{\rho u_i u_j} \right) \quad (2)$$

$$\frac{\partial}{\partial t} (\rho e_0) + \frac{\partial}{\partial x_j} [\rho u_j e_0 + u_j p + q_j - u_i \tau_{ij}] = 0 \quad (3)$$

$$\overline{-u_i u_j} = 2\nu_T S_{ij} \quad (4)$$

Where (1) is Continuity Equation, (2) is Reynolds Averaged Navier-Stokes equations, (3) is conservation of energy equation and (4) is Spalart-Allmaras turbulence model.

Using the approach from [1] the given above equations are converted to algebraic equations. It then solves for our six variables at each of the cell centers of our mesh. This means that if we have 300 000 cells, Fluent is going to solve 1.8 million equations to solve the problem, [1].

Using the approach developed at [1] for verification of the numerical results it is need to make some hand calculations.

In this numerical simulation of the flow it is expected to see many flow features in three dimensions. The following features are of interest: *Suction peak* (low pressure zone) that forms on the wing and how the size of the *suction peak* changes spanwise.

- *Shock along the wing surface* (because of the transonic flow)
- *Trailing edge vortices* (forming downstream of the wing, due to the interaction of the high and low pressure zones)
- *Lift coefficient of the wing* - c_L
- *Drag coefficient of the wing* - c_D

In the pre-analysis section an attempt has been made to predict c_L . The purpose here is to find the *lift curve slope*. The airfoil that is used for the wing, [1] is symmetrical, hence at 0 angle of attack - α , it produces no lift. The airfoil that is used here using the algorithm of [1] is ONERA OA206 Airfoil, [3].

First it has to be known the aspect ratio - AR of the wing, and it is calculated using the formula:

$$AR = \frac{b^2}{S} \quad (5)$$

Where b is the span and S is the planform area of the wing. The $AR = 3.8$. The slope of an infinite wing is calculated by the formula:

$$a_o = c_L = \frac{L}{q_\infty S} \quad (6)$$

where $q_\infty = \frac{\rho V^2}{2}$ is velocity head (dynamic pressure), and L is a lift force. The slope is $a_o = .0884$. Citing [1] then it can be calculated what lift curve slope - $a = c_L^\alpha$ for the finite wing will be using the correction for a swept wing. According to [1] it is need to use this correction for a swept wing since the free stream Mach number - M is not seen by the entire wing and instead the wing sees a lesser M , delaying the onset of a shock and increasing the critical M .

The lift curve slope is calculated by, [4]:

$$a = c_L^\alpha = \frac{a_o}{1 + \frac{a_o}{\pi AR}(1 + \tau)} \quad (7)$$

Where τ is the correction for the swept wing depending on the AR and η -wing's narrowing.

After that the calculated value of a corrected slope is $a = c_L^\alpha = .0760$.

It is known that for the finite wing, lift curve will be lower than for infinite wing due to the 3D effects and this is reflected the numerical simulation in the presented study. Once the slope for the finite wing is known, it can be calculated the c_L for the wing. Since the airfoil is symmetrical, $\alpha_L = 0$. Then the lift coefficient is calculated by:

$$c_{l_0} = a(\alpha - \alpha_L) \quad (8)$$

The $c_{l_0} = .2328$. Using the approach in [1], the calculated lift coefficient is for the entire wing, but here it is used only a half for the numerical simulation of the low speed incompressible flow. This means that there is need to use a correction for the compressibility at $M = .8395$, [1]. Hence the lift coefficient is calculated using:

$$c_l = \frac{c_{l_0}}{\sqrt{1 - M_\infty^2}} \quad (9)$$

Using formula (9), $c_l = .4284$ for the entire wing for the half wing, $c_l = .2141$.

This is approximated value of c_l by handout calculation, the expectations here is Fluent will give something comparable but less than what it was predicted because of the presents of the shock on the wing surface.

3. NUMERICAL MODEL

3.1 Geometry

Quoting [5] the ONERA M6 wing is a swept, semi-span wing with no twist. It uses a symmetric airfoil using the ONERA D section. The wing geometry is a scaled down version matching the geometry from NASA rather than the experiment available at [2]. The half span dimension is 304.8 mm and from there it was calculated the scaling factor for the entire wing. The Table 2 describes some key geometry, the leading and trailing edge angles are measured from vertical.

Table 2 Key geometry parameters

Span (mm)	Taper Ratio	Mean Aerodynamic Chord (mm)	Leading Edge Angles (degrees)	Trailing Edge Angles (degrees)
304.8	.562	164.592	30	15.8

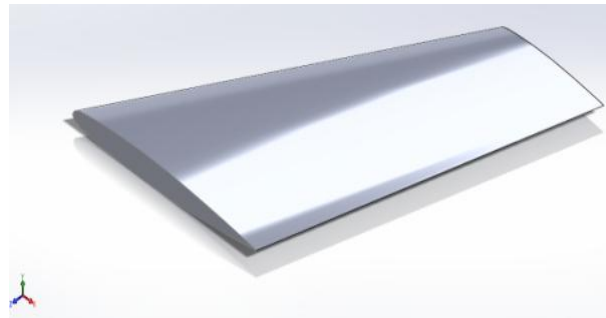


FIG.1. Wing Geometry, [1]

It is used the geometry developed in [1] and it is exported as file in ANSYS.

3.2 Boundary Conditions

Figure 2 shows the geometry and the domain around the wing in ANSYS and also the chosen boundary conditions according to [1].

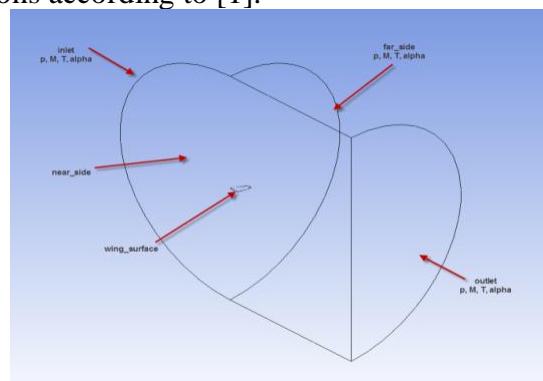


FIG.2. Geometry and fluid domain in ANSYS

Table 3 represent chosen boundary conditions and type according to [1].

Table 3 Boundary Conditions

Boundary Conditions	Boundary Type	Condition
Inlet, Far_side, outlet	pressure far-field	$p = 45.8290 \text{ psi}$ $T = 460R$ $M = .8395$ $\alpha = 3.06^{\circ}$
near_side	symmetry	symmetrical boundary
wing_surface	wall	$\nu = 0$

3.3 Mesh

Creating an accurate mesh involves a grid generation using appropriate shape cells; here the cell for the mesh is triangular. Essentially, it consists of converting the grid into a format which can be understood by the Fluent solver in order to approximate the equations of the fluid mechanics in each cell. Figure 3 shows the generate mesh around the wing and the number of cells.

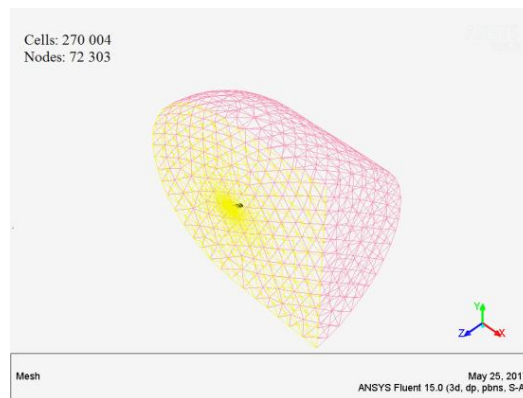


FIG.3. Mesh part in ANSYS Fluent with cells and nodes

4 NUMERICAL RESULTS

4.1 Lift and Drag coefficient

After the simulation is done, Fluent give the following values about c_l and c_d coefficients:

$$c_l = 0.11423573 \quad (10)$$

$$c_d = 0.013593919 \quad (11)$$

4.2 Pressure and Mach number distribution.

Pressure distribution

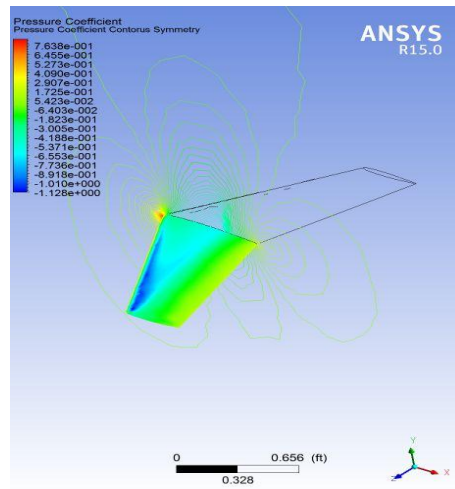


FIG.4. Pressure coefficient Contours Symmetry

Fig. 4 shows the pressure coefficient contours symmetry over the wing. It can be notice from Fig. 4 on the left side of half wing forming the shock.

After changing the view into Oxy plane, Fig. 5 it can be seen to the upper surface of the wing thin boundary layer and its thickness after the shock.

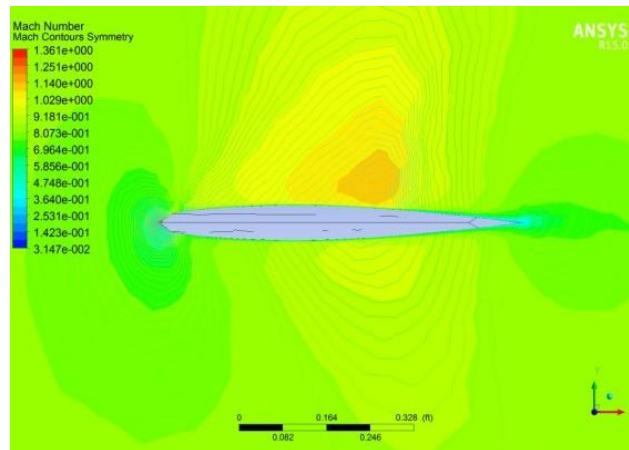


FIG.5. Forming of the boundary layer

Mach number distribution

Looking at Fig. 6 you can see where the shock was compared that to the Mach number. Quoting [8] at higher speeds, as the advancing tip Mach number approaches 1.0, its lift becomes restricted by shock-induced flow separation leading to drag.

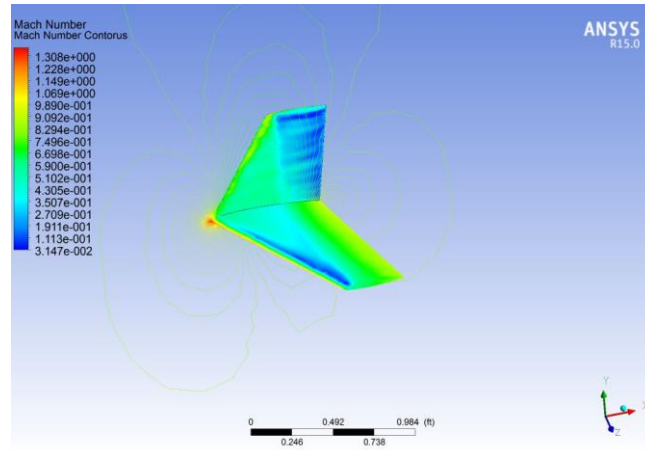


FIG.6. Mach number

4.3 Trailing Edge Vortices.

Looking at Fig. 7 it is easy to see trailing edge velocity vectors. Fig. 7 illustrated velocity vectors at the trailing edge and it can be looked along the way and the existence of trailing edge. Cities source [1] it is deduced that trailing edge vortices or lift induced vortices are a really important phenomenon when it comes to finite wings. These vortices are formed because of the finite length of the wing. The high and low pressure regions interact with one another at the wingtips and this interaction creates the vortices trailing downstream of the wing. Cited [7] the phenomenon of wake vortices is particularly dangerous in an airport because the vortices generated have a high intensity and that could lead to crashes when a plane is about to land or to take off.

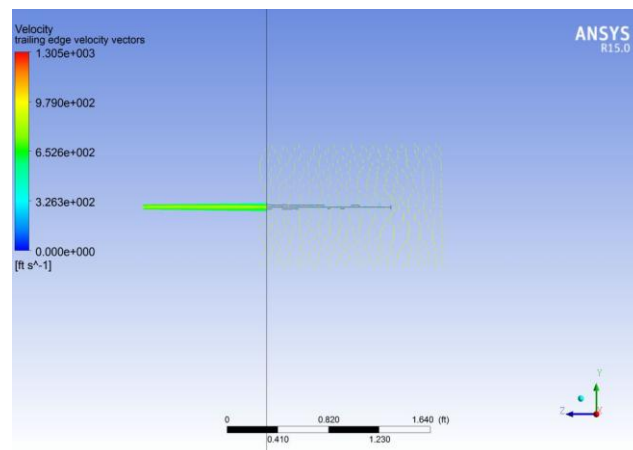


FIG.7. Trailing Edge Velocity Vectors

4.4 Plotting and Comparing the Pressure Coefficient.

Fig. 8 shows the chart of pressure coefficient - c_p and gives the results for c_p from Fluent.

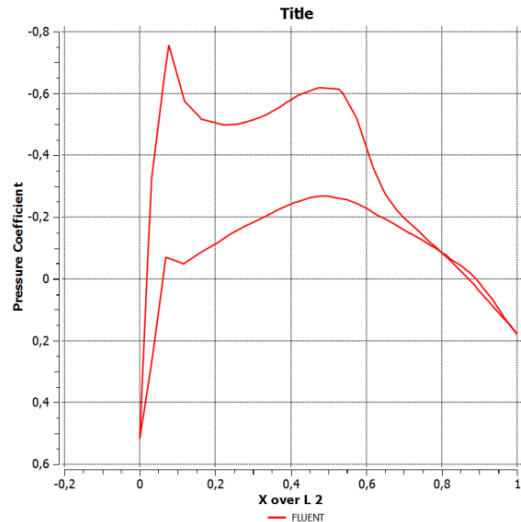


FIG.8. Chart of C_p from Fluent

Fig. 9 illustrates the comparison that was made between c_p results from Fluent and experimental data, taken from [2] citing [1].

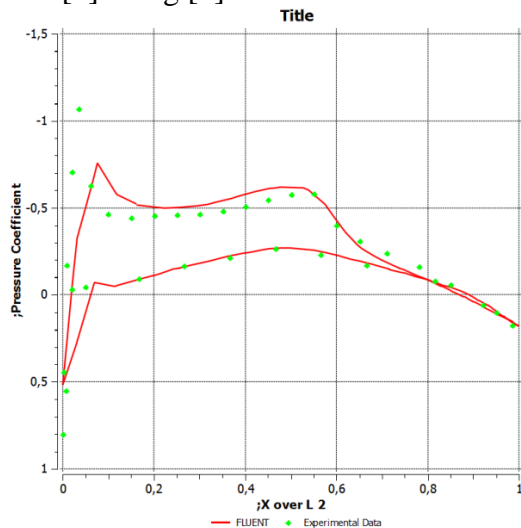


FIG.9. Comparison between C_p from Fluent and experiment

5. VERIFICATION OF OBTAINED NUMERICAL RESULTS

Citing [1] the obtained numerical results can be verified by looking at mass conservation and by comparing CFD results with those available from NASA.

At this study are used two criteria for validation and verification of the results:

- 1) Comparison of developed mesh for pressure parameter with both NASA and Fluent.
- 2) Comparison of aerodynamic coefficient of the wing c_l and c_d obtained with original mesh and NASA CFD, [2].

Fig. 10 shows the made comparison of developed mesh both with NASA and Fluent.

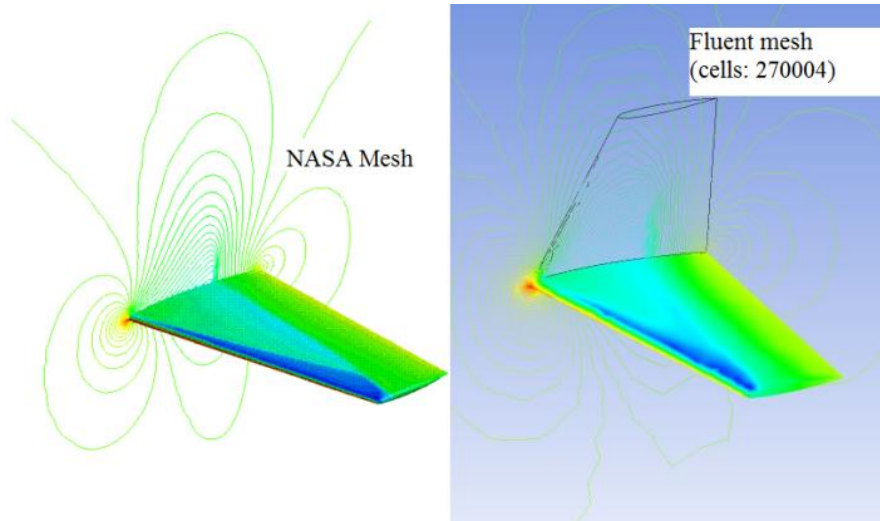


FIG.10. Mesh comparison

It can be noticed from the figure that the NASA mesh is too fine and it is clear from the streamlines in the meantime developed here Fluent mesh is consisting of 270 004 cells and it is coarse which respectively shall reflect on the results. Consequently for more accuracy it should be generated by the user more fine mesh or to use refinement techniques of Fluent.

Using the approach developed in [1] and applied it here into this study it is made a comparison of c_l and c_d obtained with original mesh and NASA CFD, [2], Table 4.

Table 4

	c_l	c_d	% difference of c_l	% difference of c_d
NASA CFD	.1410	.0088	-	-
Original mesh	0.11423573	0.013593919	23%	54%
Hand calculation	.2141		46%	

6. CONCLUSION

The main conclusion that can be made onto this stage of the study is that observing the results for lift and drag coefficients shown at Table 4, the calculated percentage errors for both values are too big and it is due to the coarse mesh.

For more precision it have to be made more numerical experiments and it have to be included the technique mesh refinement.

REFERENCES

- [1] Cornell University “MAE 5230”, Aug 10, 2015;
- [2] NASA’s Glenn Research Center validation archive: Slater, John W., “ONERA M6 Wing Study #1.” *ONERA M6 Wing*. NPARC, n.d. Web;
- [3] <http://airfoiltools.com/airfoil/details?airfoil=oa206-il>;
- [4] Гешев Д., *Аеродинамика на Летателните апарати*, Катедра „Въздушен транспорт“, ТУ-София 2002;
- [5] <https://www.grc.nasa.gov/WWW/wind/valid/m6wing/m6wing.html>;

- [6] Rotaru C., Circiu I., Boscoianu M., *Computational Methods For The Aerodynamic Design*, Review of Air Force Academy, ISSN: 2069-4733, ISSN-L: 1842-9238, pp.43-49, 2(17) 2010;
- [7] C. E. Moldoveanu, P. Şomoiaş, O. Hemelaar, M. Aubert, *Particularities of the Interaction Between the Aircraft Wake Vortices and the Atmospheric Boundary Layer*, Review of the Air Force Academy, DOI: 10.19062/1842-9238.2015.13.3.6, pp. 95-100, 2015;
- [8] C. Rotaru, R. I. Edu, *Lift Capability Prediction for Aerodynamic Configurations*, Review of the Air Force Academy, ISSN: 1842-9238, pp. 57-62, 2014.